



Book Name: *Part Modeling User's Guide*

Creating Features

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This chapter provides basic information on how to create protrusions, cuts, slots, and other sketched solid features. It describes common aspects, such as feature form options and techniques of three-dimensional sketching. These aspects are initiated when you choose Feature from the Part menu, Create from the Feat menu, and Solid from the Feat Class menu. For detailed feature-specific information, see the appropriate sections in Construction Features, Surface Features, and Freeform Manipulation.

Topic

Sketching on a Part

Feature Form Options

Thin Features

Extrude

Revolve

Sweep

Blend

Advanced Form Features

Creating a Merge Feature

Retrieving Pro/DESIGNER Data

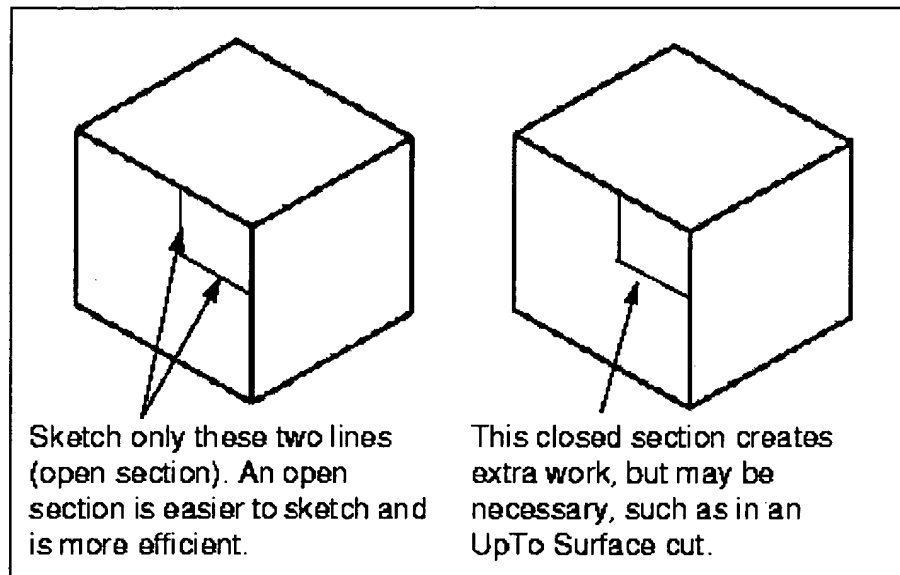
Sketching on a Part

Certain sketcher functionality is available only when you sketch on a part. These options, and special implementations of previously presented functionality, are described in the sections that follow.

When you sketch sections on a part, sketch only what is necessary to do the job. If you do not want to create a new surface, do not sketch a line in the section. For example, when sketching the corner cut as shown in the illustration Sketching on a Part, sketch the open section as shown, and not the closed section. If Pro/ENGINEER has problems intersecting the feature with the part, you must close the section.

When sketching, you can either reorient your model so the sketching plane is parallel to the screen, or you can stay in the same 3D orientation. For more information, see Sketching in 3D.

Sketching on a Part



Setting Up the Sketching Plane

All sections are created on two-dimensional planes. Therefore, when you sketch on a three-dimensional part, you must define the sketching plane.

How to set up the sketching plane

1. Select the sketching plane (this is usually the surface from which the feature extends).
2. Select a direction in which to extend the feature, or the direction of viewing the sketching plane.
3. Select a horizontal or vertical reference plane that orients the sketching view.

Selecting the Sketching Plane

You can select a datum plane or a planar surface as the sketching plane. Optionally, you can create a datum plane "on-the-fly" on which to sketch. The sketching plane is infinitely large.

How to specify the sketching plane

1. When the SETUP SK PLN menu appears, choose one of the following options:
 - **Use Prev**-Use the sketching plane and orientation of the previous sketch.
 - **Setup New**-Select or create a sketching plane and define its orientation. Choose an option from the SETUP PLANE menu:
 - **Plane**-Pick an existing planar surface or datum.
 - **Make Datum**-Create a datum plane for temporary use as a reference. See "[Datums](#)", for information on creating datum planes.
2. Once you have indicated the sketching plane, define the feature direction, as described in [Selecting](#)

Feature Direction, and orient the sketching plane, as described in Specifying a Horizontal or Vertical Reference.

It is possible to create several datum planes on-the-fly and use the last one created as the sketching plane. To make such a chain of internal datums, create the first one by using Make Datum. Choose Setup New again and then choose Make Datum. The plane that you previously created on-the-fly is then available as a reference for the current one. You can use Make Datum repeatedly without Setup New, however, only the plane that you create or select immediately after the last Setup New will be used for the sketching plane.

Selecting Feature Direction

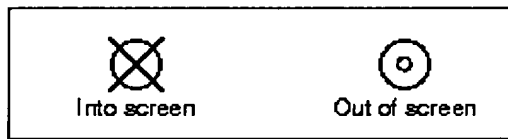
After you set up the sketching plane, Pro/ENGINEER prompts you to specify in which direction the feature should extend, or the direction of viewing the sketching plane.

How to specify the direction of feature creation

1. Pro/ENGINEER displays a red arrow on the selected sketching plane to indicate the default direction of feature creation. The appearance of the arrow depends on the part orientation.

If the part is oriented such that the feature will be created "into" the screen, the arrow points directly into the screen with the "feathered end" closest to you. If the part is oriented such that the feature will be created "out of" the screen, the arrow points out of the screen with the rounded "head end" towards you. In any other orientation, you will see the arrow from the side.

Direction Arrows

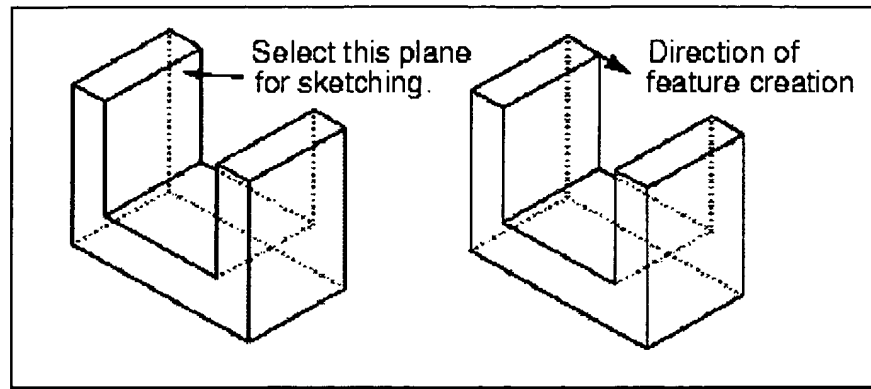


2. Pro/ENGINEER also displays the ARROW FLIP menu. Choose **Flip** to reverse the direction of feature creation, or **Okay** to accept the direction shown.

Note:

To reset the size of the flip arrow, set the configuration option "flip_arrow_scale". The default size is 1.

Selecting the Sketching Plane and Feature Direction



After you specify the reference plane, the system orients the sketching plane so it is parallel to the screen. If you are creating a feature that adds material to the part (such as a protrusion), the system orients the plane so the feature "grows" towards you. If you are adding a feature that removes material (such as a slot), Pro/ENGINEER orients the plane so the feature extends away from you.

Specifying a Horizontal or Vertical Reference

To fully specify the sketching view, you need to orient the sketching plane to the screen normal axis. You do this by specifying a horizontal or vertical reference plane. This plane must be perpendicular to the sketching plane. Choose Top, Bottom, Right, or Left from the Sket View menu and pick on the plane you want to face towards the specified side of the screen.

You can select an edge as a Top, Bottom, Left, or Right reference for the sketching plane. The selected edge must lie on a plane that is perpendicular to the sketching plane.

The default value for the Sket View menu is the previous selection. The starting default value for the session is Bottom.

If you are creating a feature that is not aligned with the existing edges of the part, you may want to create a datum plane as a horizontal or vertical reference that will be added (that is, an "ang_plane" datum). This is especially useful when you create radial feature patterns (see [Patterning Features](#) for information about creating patterns). When you create a datum plane as a reference, first select the direction you want the plane to represent (Top, Bottom, and so on), then create it. The yellow side of the datum plane will face towards the specified side of the screen.

Sketching in 3D

The SketStart2D option in the Environment menu controls whether Sketcher reorients the solid object when it starts up. When this option is set (the default), Sketcher makes the sketching plane parallel to the screen. If this option is unset, Sketcher does not reorient the solid object when it starts. SketStart2D is effective only if the model already contains some geometry.

Use the Sketch View option in the Sketcher menu to reorient the model so the sketching plane is parallel to the screen.

You can use the "sketcher_starts_in_2d" configuration option to set the starting value of the SketStart2D option. Use the new "sketcher_starts_in_2d" option instead of the old "sketch_in_3d" option.

The "sketcher_starts_in_2d" option has the following settings:

- "yes"-(Default) Start Sketcher in 2D orientation. When Sketcher starts, the system reorients the model so the sketching plane is parallel to the screen.
- "no"-Do not change the orientation of the model so you can sketch in the same 3D orientation.

Entering the Sketcher Environment

Sketcher Grid

When you enter Sketcher, the system displays the grid and enables grid snap for any sketching plane orientation except "edge-on." If you select a new origin for the grid, the system projects that point onto the sketching plane to determine the new origin.

Note:

Only Cartesian grid is supported for sketching in 3D.

Retrieving Existing Sections

The Place Section option in the Sec Tools menu allows you to retrieve a section from disk or from memory and place it on the current sketch as an independent copy of the original section. The target section can be empty or can contain existing entities (and dimensions). The retrieved entities and their dimensions are added to any existing section data. In a parallel blend, the retrieved section will be added to the current subsection.

The Place Section option copies the entities and relations (if any) of the original section without reference to the original context in which they were created. Thus, the accuracy and units of measure are those of the current model. The placed section will behave as if you had created it directly using Sketcher commands.

Because the placed section is independent of the original section, a change to the placed section will not cause a change to the original section. This allows you to read the unchanged retrieved section several times into the current section or another feature. For parallel blends, you can place the read section into different subsections with variations in rotation angle and size. In addition, the placed section will not reflect any changes to the original section after the system performs the Place Section command.

For the first feature, if you are placing the section into a sketch that has existing entities, Pro/ENGINEER displays a small subwindow to help you place the section on the currently active model, and prompts you to do the following:

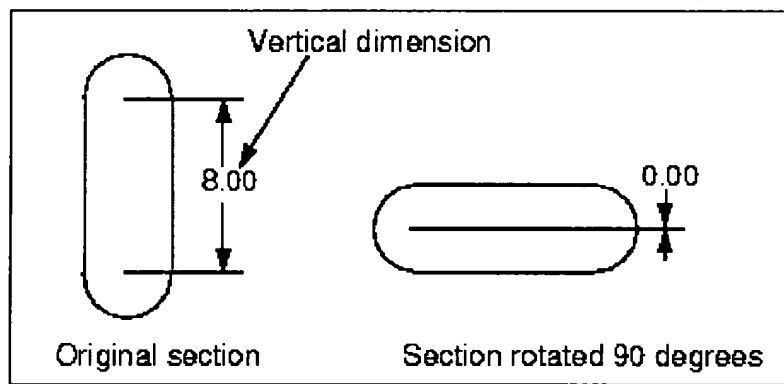
- Enter an angle, if desired, by which the section in the subwindow will be rotated.
- Select an origin point on the entity in the subwindow for scaling the placed section.
- For placement purposes, select a point on the entity in the subwindow that corresponds to a red circular mouse cursor in the Sketcher window. You can drag this point for moving or changing the scale factor.

- Specify an initial scale factor.

How to retrieve an existing section

1. Choose **Place Section** from the SEC TOOLS menu. Pro/ENGINEER prompts you for the name of an existing section. Enter a section name, or list the available sections by entering a [?]. Once you specify a name, the system retrieves the section and displays it in a subwindow.
2. If the section sketch is being placed on a part sketching plane (*not* an auxiliary sketch, such as for a sketched blind hole or a shaft), you can modify the location, orientation, and scaling of the section. For these actions, continue with Step 3.
3. Enter a rotation angle for the sketch. Be aware that some dimensioning schemes may change because of the change of sketch orientation, as shown in the following illustration.

Effect of Sketch Orientation on Dimensions



4. Select an origin point on the sketch for scaling. When scaling the section using the mouse, this origin point remains stationary.
5. Select a drag point on the sketch. This is the point that will follow the mouse during positioning. The drag point cannot be coincident to the scaling point.
6. Enter a preliminary scale factor for the sketch.
7. Now the section can be placed on your part. Move the mouse from the subwindow to your part window. The section appears in red and follows your mouse pointer as it moves around the screen.
8. Using the mouse, you can do any of the following:
 - Click the left button to place the section. The section changes from red to the normal section color and the system displays any dimensions.
 - Click the middle button to abort the section placement and return you to step 3.
 - Click the right mouse button to toggle back and forth between scaling and drag modes. When scaling, your scale origin remains stationary and moving the mouse increases or decreases the size of the section. Returning to drag mode causes the drag origin to follow the mouse again.

9. Locate the section with respect to the part by dimensioning or aligning.
10. Regenerate the section.

Restarting the Sketch

The Delete All option in the Deletion menu lets you delete all geometry in the section, similar to erasing everything off a sheet. Delete All affects only entities and dimensions and does not undo changes made to the Sketcher environment.

When you are in redefine mode, Delete All only deletes all dimensions.

Creating Sketch Geometry

Once the sketching plane is defined, Pro/ENGINEER displays the Sketcher menu, and automatically selects the Sketch option in the Section menu.

When sketching on a part, you use the same techniques as in 2D Sketcher mode (see [Sketcher](#)), and also additional tools, detailed in the following sections.

Note:

If you have the Pro/PIPING option, you can use the centerlines of pipe segments as references in the 3D Sketcher just as you use curves in the following descriptions.

The following topics describe the 3D Sketcher options:

- **Use Edge** (see [4 - 9](#))
- **Offset Edge** (see [4 - 12](#))
- **Pick Curve** (see [4 - 19](#))

The Use Edge Option

The Use Edge option in the Geom Tools menu duplicates and automatically aligns certain geometry. This is especially useful for duplicating splines in non-parallel planes. The Use Edge option creates sketched entities from model geometry by copying the edge definition onto the sketching plane. This geometry can be treated like any other sketched geometry, and will automatically be aligned to the geometry of the part. In this process, the endpoints of the entity will also be aligned to the endpoints of the edge. The Use Edge option uses each edge "as is". For example, if an edge has been trimmed by a cut, a second cut using the trimmed edge will become a child of the first because the location of one of the new edge endpoints is defined by the first cut.

When you use the Use Edge option, you can have the model oriented any way that is convenient (see the illustration [Creating a Section Using Use Edge and Sketched Entities](#)).

In Sketcher mode, the Use Edge option allows you to pick an existing part axis to create a centerline that is automatically aligned to the axis.

Note the following restrictions:

- 360° circles are broken into two edges. You must select each edge separately by the Use Edge option.
- You cannot directly select composite datum curves for Use Edge. Instead, use Query Sel to select the underlying simple curves.
- A spline silhouette edge is not selectable for the Use Edge operation.

How to use the Use Edge option

1. Choose **Use Edge** from the GEOM TOOLS menu.
2. To create sketched entities offset from a single edge, choose **Sel Edge** from the USE EDGE menu.

To create sketched entities from a loop of edges or entities, choose **Sel Loop** from the USE EDGE menu. Select a face containing the edges or entities. If more than one loop is possible, use **Next** and **Previous** from the CHOOSE menu to select the desired loop.

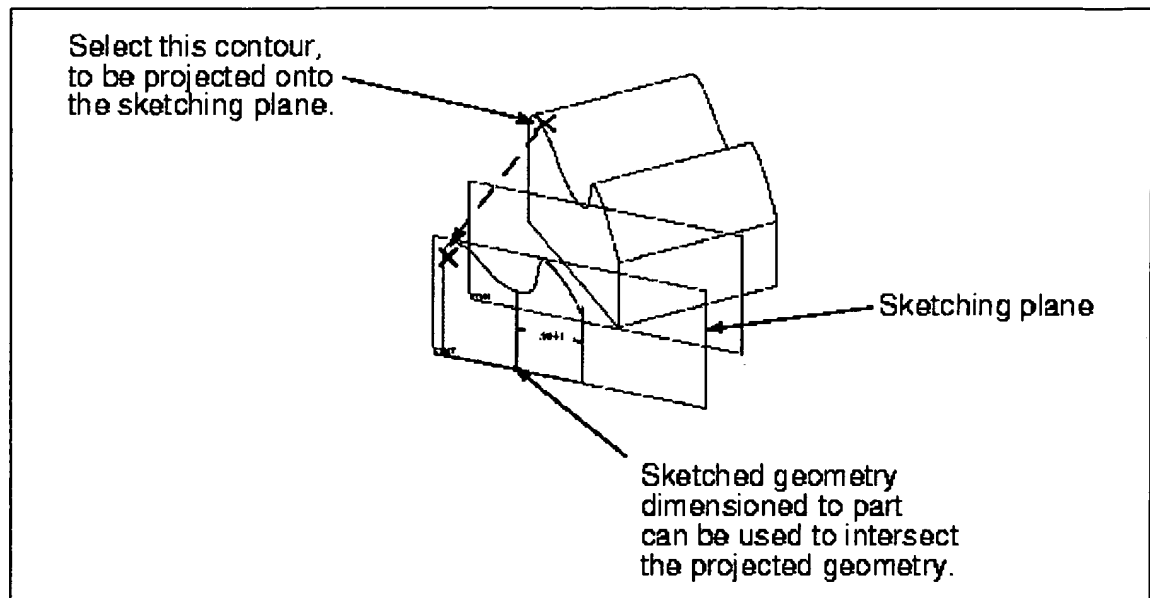
To create sketched entities from a chain of edges or entities, choose **Sel Chain** from the USE EDGE menu. Select the beginning and ending entities of the chain. If you select curves, they must both belong to the same datum curve. If you select edges, they must belong to the same surface or face. You can pick two edges on a part's geometry or two one-sided edges of a quilt.

With **Sel Chain**, if you pick two entities that belong to an IGES wireframe or a datum curve in a uniquely defined plane, the Sketcher tries to choose a chain that connects the entities and lies in that plane.

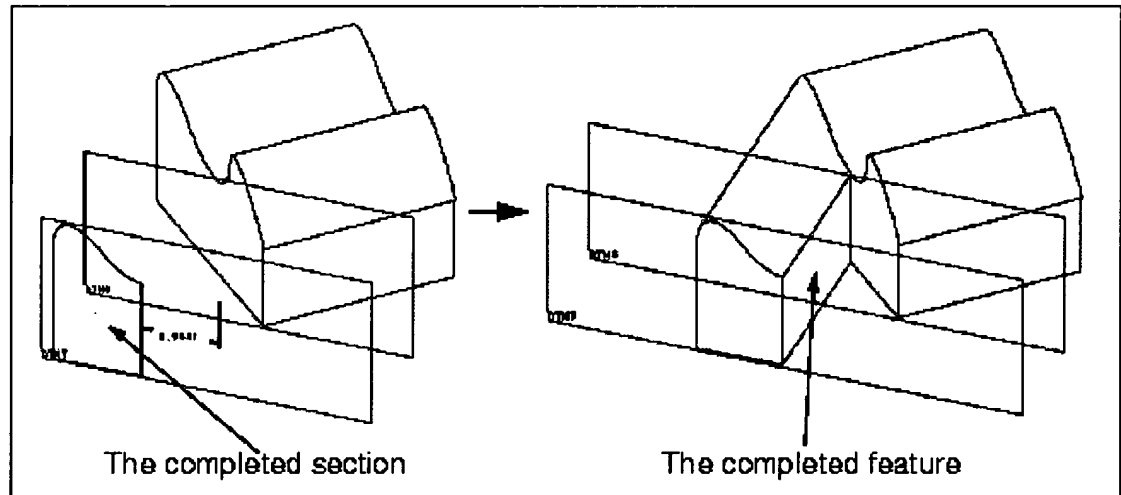
If more than one chain is possible, use **Next** and **Previous** from the CHOOSE menu to select the desired chain.

3. The selected geometry highlights temporarily in blue, then in sketching color (cyan or white). Once the process has been completed, Pro/ENGINEER displays an appropriate message in the Message Window.
4. Use the Sketcher to add entities where desired. Use the **Intersect** option from the GEOM TOOLS menu to modify the entities created using **Use Edge**. Dimension the sketched geometry appropriately.
5. Continue creating the feature in a normal manner.

Creating a Section Using Use Edge and Sketched Entities



Completed Feature Created with Use Edge



The Offset Edge Option

The Offset Edge option in the Geom Tools menu simplifies the duplication of geometry. This geometry can be treated like any other sketched geometry. You can create offset entities from edges that are lines, arcs, or splines. When you create an offset entity, each point of the original lines, arcs, or splines is first projected onto the sketching plane. Each point is then offset normal to the projected entities by the specified distance. For example, creating an offset arc results in a concentric arc of a different diameter, rather than in a translated copy of the same arc.

The Offset Edge option offsets each edge "as is." For example, if an edge has been trimmed by a cut, the location of one of the new edge endpoints is defined by the cut. This endpoint will be used in the offset operation.

Offset edges can be created using a single entire edge (untrimmed), a portion of a single edge (trimmed), a chain of edges or entities, or a loop of edges or entities.

Note the following restrictions:

- 360° circles are broken into two edges. Each edge is selected separately by the Offset Edge option.
- You cannot offset edges that have tangency that meets in a sharpened point.
- When you select tangent edges to offset, select them all at the same time using the Sel Chain option. Otherwise, the section will fail regeneration because the individual offsets of the tangent entities are incompatible.

Invalid Tangency for Offset Edge

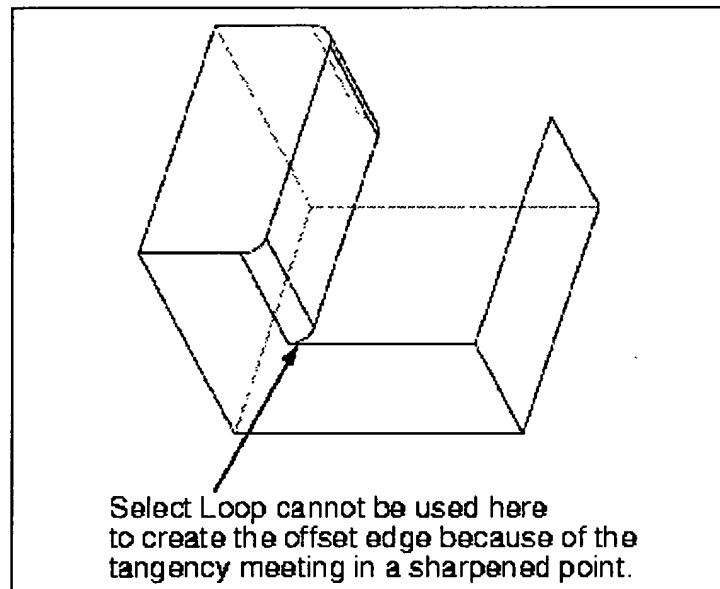
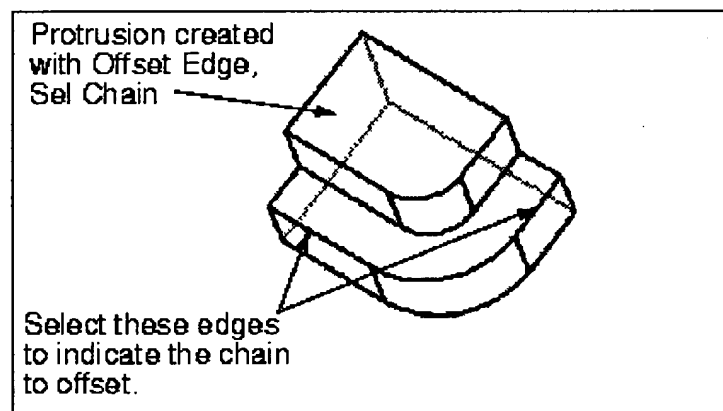


Figure Created with Offset Edge, Sel Chain



The following sections describe how to use the Offset Edge option to offset an entire edge, a portion of an edge, a chain, and a loop.

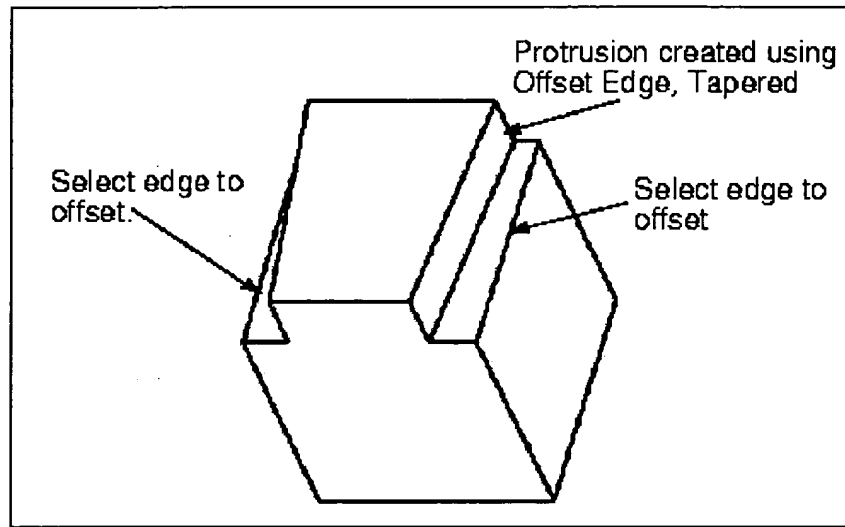
Using Offset Edge to Offset an Entire Edge

To use the whole edge to create an offset entity, you must create an *untrimmed* offset edge from a single edge.

How to create an untrimmed Offset Edge from a Single Edge

1. Choose **Geom Tools** from the SKETCHER menu.
2. Choose **Offset Edge** from the GEOM TOOLS menu.
3. Choose **Sel Edge** from the OFFSET SEL menu, or **Sel Loop** or **Sel Chain**, if desired.
4. Select the desired edge. You can select two edges on a part's geometry, or two one-sided edges of a quilt.
5. If you chose **Sel Edge**, choose one of the options in the OFFSET TYPE menu:
 - **Fixed**-Create an entity with fixed offset at any point.
 - **Tapered**-Create an entity with different offsets for each endpoint.
6. Choose **Untrimmed** from the OFFSET TYPE menu.
7. Choose **Done**.
8. If you chose **Fixed**, Pro/ENGINEER displays a red arrow near the middle of the edge. Enter an offset in the indicated direction. If you chose **Tapered**, the system displays the arrow and the prompt for both endpoints of the edge.
9. Once you have entered the values, Pro/ENGINEER creates the entity, with the offset dimension shown in symbolic form. Once the process has been completed, the system displays an appropriate message in the Message Window.
10. Use the Sketcher to add entities where desired. Use the **Intersect** option from the GEOM TOOLS menu to modify the entities created using **Offset Edge**. Dimension the sketched geometry appropriately.
11. Continue creating the feature in a normal manner.

Feature Created with Untrimmed Offset Edge, Tapered



When you delete an offset edge, Pro/ENGINEER retains the corresponding known entities (see the section Dimensioning Sections to a Part) to serve as alignment references. To delete these known entities, delete the offset dimensions.

Using Offset Edge to Offset a Portion of an Edge

To use a portion of the edge to create an offset entity, you must create a *trimmed* offset edge from a single edge.

How to create a trimmed Offset Edge from a Single Edge

1. Use the **Point** option from the GEOMETRY menu to place points on the edge at the locations to which you will trim *before* you create entities offset from the edge.
2. Choose **Geom Tools** from the SKETCHER menu.
3. Choose **Offset Edge** from the GEOM TOOLS menu.
4. Select the desired edge.
5. Choose either **Fixed** or **Tapered** from the OFFSET TYPE menu.
6. Choose the OFFSET TYPE menu option **Trimmed**. This option requires that you have first placed points on the edge at the locations to which you will trim the edge.
7. Choose **Done**.
8. The system prompts you to select a point to trim the edge. Pick one of the points.
9. Enter the offset value in the indicated direction.
10. Select the second point. If you chose **Tapered**, enter the offset value for the second endpoint.
11. Pro/ENGINEER creates the offset entity. Continue to complete the feature (see Steps 8 to 10 for the untrimmed edge).

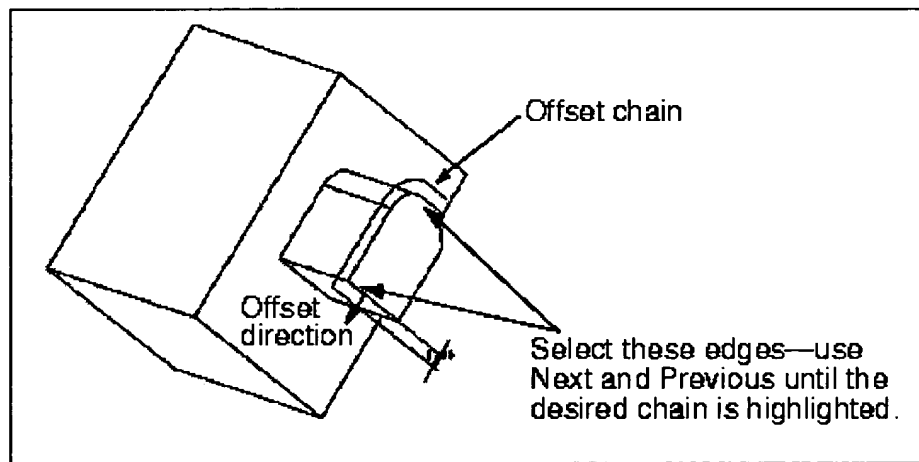
Using Offset Edge to Offset a Chain

How to offset a chain of edges or curves

1. Choose **Geom Tools** from the SKETCHER menu.
2. Choose **Offset Edge** from the GEOM TOOLS menu.
3. Choose **Sel Chain** from the OFFSET SEL menu.
4. Select the beginning and ending entities of the chain to be offset. If you select curves, they must both belong to the same datum curve. If you select edges, they must belong to the same surface or face.
5. If more than one chain is possible, use **Next** and **Previous** from the CHOOSE menu to select the desired chain.
6. Enter the offset value. A red arrow indicates the default direction of the offset. To offset in the opposite direction, enter a negative value.

Pro/ENGINEER offsets the whole chain in the same direction. The entities are extended and trimmed, as necessary to remain connected.

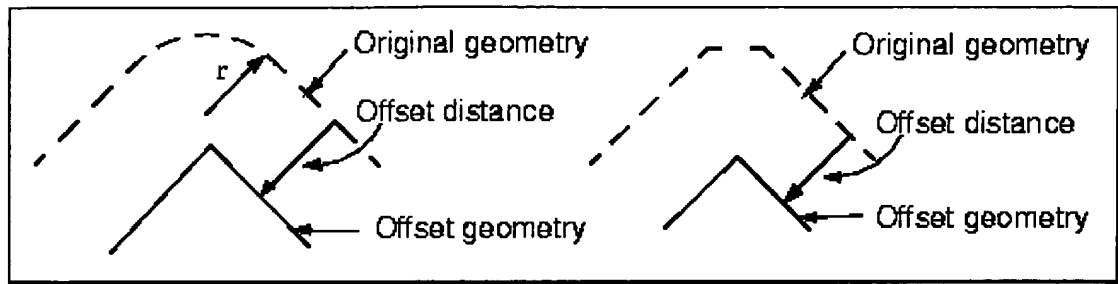
Offsetting a Chain



When you use Offset Edge, Select Chain to offset a chain of entities by a large distance, the system creates offset geometry according to the following guidelines:

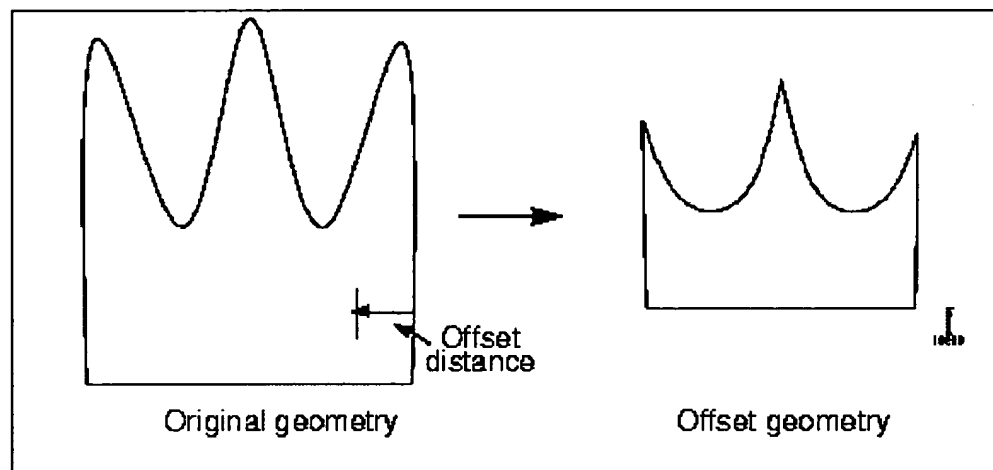
- If the offset is inward along an arc and the offset value exceeds the radius of the arc, the system removes the arc from the offset chain. Similarly, the system might remove other entities because of excessive offset value. If you lower the offset value later, the entities reappear. The following figure illustrates such a case.

Offsetting a Curve



- If the offset value is more than the local minimum radius of curvature for entities that compose the offset chain, the system creates an offset chain that might have a different number of entities. In the following example, offsetting a spline by a large value causes the resulting spline to be broken into several pieces. If the offset value is changed, the system can "piece" together the broken spline so it becomes a single entity again.

Offsetting a Spline



Using Offset Edge to Offset a Loop

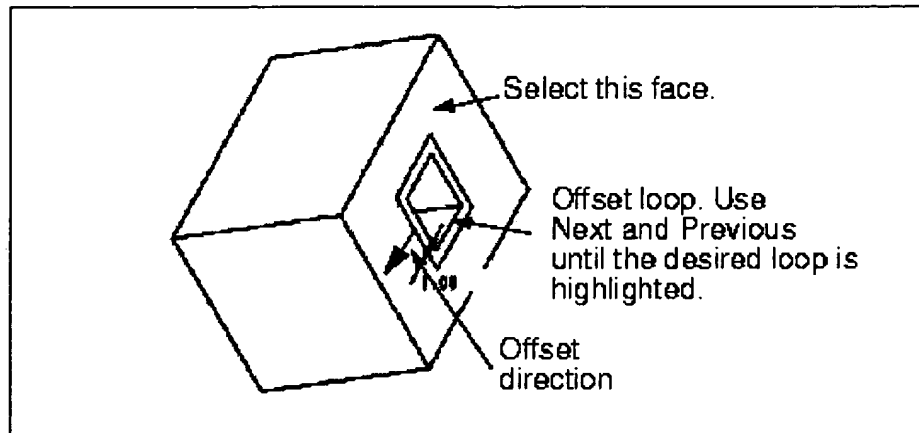
How to offset a loop of edges or entities

1. Choose **Geom Tools** from the SKETCHER menu.
2. Choose **Offset Edge** from the GEOM TOOLS menu.
3. Choose **Sel Loop** from the OFFSET SEL menu.
4. Select a face containing the edges or entities to offset.
5. If more than one loop is possible, use **Next** and **Previous** from the CHOOSE menu to select the desired loop.
6. Enter the offset value. A red arrow indicates the default direction of the offset. To offset in the opposite direction, enter a negative value.

Pro/ENGINEER offsets the whole loop in the same direction (see the following figure). The entities will be stretched and squeezed as necessary to remain connected. A single dimension will be created

automatically between the loop and the original entities.

Sample of Offsetting a Loop



The Pick Curve Option

The Pick Curve option is available only in a special 3D Sketcher mode with no sketching plane assigned, such as when you create composite datum curves. This option enables you to select edges or three-dimensional curves (similar to Use Edge in regular Sketcher mode). You can pick two edges on the geometry of a part, or two one-sided edges of a quilt. For each entity selected, Pro/ENGINEER creates a three-dimensional sketched entity (displayed in cyan) directly on top of it.

The system displays the Crv Sketcher menu with the options described in the section [Processing with the From Crv Option](#).

Note:

You cannot import any IGES features into the 3D Sketcher.

Dimensioning Sections to a Part

Sections sketched on a part must be dimensioned for size and relative placement on the part. Dimensioning can occur with the model in any orientation; the sketching plane does not have to be parallel to the screen. In fact, dimensioning the section with the model in an iso-type view sometimes helps avoid invalid dimensioning (see the illustration [Dimensioning to Part Edges](#)).

When sketching on a part, you can create three types of dimensions using the Dimension menu:

- **Normal**-Create a dimension that references sketched entities only, or between a sketched entity and part geometry. These dimensions are used to solve the section and eventually become regular part dimensions.
- **Known**-Create a dimension that references *part geometry only*. These dimensions are used to drive Sketcher dimensions through a relation; they are necessary to solve the section, and are not displayed anywhere except in this section. Known dimensions have the symbolic form "*kd#*".
- **Baseline**-Establish a baseline for ordinate dimensioning. See [Ordinate Dimensions](#) for more

information.

Aligning or Dimensioning to a Model Edge or Surface

Aligning or dimensioning to a model entity (edge, curve, and so on) adds the existing geometric entity to the list of "known" entities that Pro/ENGINEER uses with its implicit rules to solve a section. All open ends *must* be explicitly aligned to the model edges. Aligning to a model edge does not automatically move sketched geometry to be coincident with the model. Instead, the system considers the proximity of sketched geometry to aligned model edges when solving the sketch.

You can align straight lines, circular edges and circular centers, and entity endpoints. Composite curves cannot be directly referenced when aligning sketcher entities. Instead, you must align the sketcher entities to the underlying curves that make up the composite curve.

You can also align to surfaces. In Sketcher mode, surfaces are highlighted in red when you pick an edge for alignment (using the Get Select menu). If you use Query Sel, you can toggle to the edge of the surface (displayed in blue) for alignment. The system chooses the edge-on surface for aligning and dimensioning by default, rather than the edge of this surface.

There are two ways to align to an edge of a part using the Align option:

- You can select the model geometry to add to the list of known entities before doing any sketching. To do this, select the model geometry with the left mouse button, then click the middle button. The system highlights the geometry and informs you that the geometry is aligned.
- You can align the sketched geometry to the model by selecting one model edge and one sketched entity.

Known entities are displayed in orange phantom font. If you align a sketcher entity to an edge or curve in a model, the edge or curve will be displayed in phantom orange.

Avoiding Implicit Alignment

When geometry is close to a model edge, it may be within the accuracy for the Sketcher to assume alignment, even though to validate the section you must explicitly align it. However, you may want sketched geometry to be close to model geometry, while still being able to maintain dimensional control on this placement. This means you want to override a Sketcher assumption. To do this, set the configuration file option "use_dimensioned_edges" to "no". This means that you do not want Pro/ENGINEER to assume any implicit alignment with edges to which features have already been aligned or dimensioned.

Rules to Remember

When you align entities, remember the following rules:

- When you align a circular edge, both the size and location of the circle are considered to be known.
- You can align splines and conics to edges as a way to establish tangency at their endpoints. The sketched conic or spline must lie close to the edge, and must be close to tangent to be aligned successfully. To align conics, select the endpoint, then pick the edge. To align splines, select the spline twice, then select the edge.

- It is possible to align to a "silhouette" edge of a cylinder, cone, or other revolved features, even though no physical edge exists there. The silhouette edges must be on the sketching plane.

Aligning to Points

You can align and dimension entities to known points of the model. Known points are vertices, datum points, and axes normal to the sketching plane (so they are projected as a point). When you sketch on a part, you can align to the known points to capture relationships between features.

The procedure of aligning to points is the same as for edges. To select a vertex, pick close to it. To select a datum point or axis, pick on the text.

You can also select the known points for dimensioning.

Unaligning Geometry

Geometry previously aligned in Sketcher mode can also be unaligned by choosing Unalign, Unalign Many, or Unalign All. The system highlights those entities that are explicitly aligned (using the Align option) in green.

Choose Unalign to remove the alignment from an individual entity, then select it. As each section entity is selected, the green highlighting disappears. If you remove the alignment from a model edge, all section entities aligned to it will be unaligned.

Unaligning Multiple Entities

The option Unalign Many simultaneously removes the alignment from several section entities. After choosing Unalign Many, use the Pick Many option in the Get Select menu to rubberband a box around the entities to be unaligned. All entities completely within the box are selected. Both Unalign Many and Pick Many can select known entities (representing projections to the sketching plane of referenced part geometry and displayed in orange phantom font). If you choose Unalign Many and Query Sel, the system prompts you with messages denoting section selections of regular or known entities, and what kind of entity was selected.

Use Unalign All to unalign all aligned entities in the sketch. When you choose this option, the system prompts you to confirm the request. When you choose Done Sel or click the middle mouse button, the selected entities will be unaligned.

When you remove the alignment an entity, you may need to add dimensions to solve the section.

Dimensioning to Part Edges

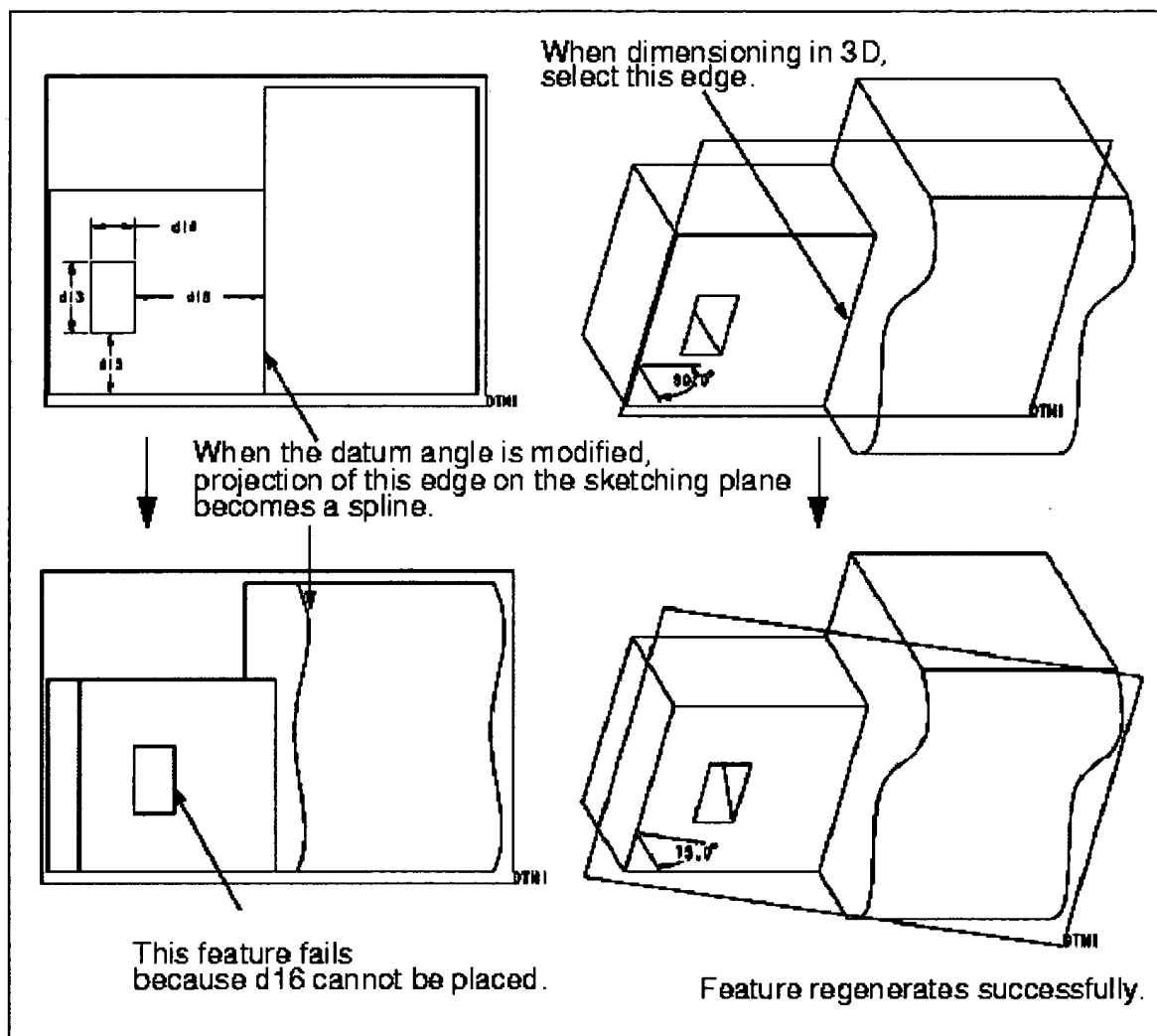
When dimensioning to a part edge, you must consider what type of entity the part edge is. Splines and arcs may appear in the sketching plane as straight lines to which you can dimension. But, if the plane that was used as the sketching plane is modified (for example, the angle of a datum plane changed), the spline or arc no longer appears as a straight line. Thus, the dimensioning scheme becomes invalid, Pro/ENGINEER will not know where to place the feature, and the feature creation or regeneration will fail.

One way to avoid this situation is to not dimension to splines or arcs that project onto the sketching plane

as straight lines. However, if the situation does occur, modify the dimensioning scheme to dimension section geometry to linear geometry. You can orient the part in an iso-type view to help you select proper geometry on the part.

The illustration Dimensioning to Part Edges illustrates how to dimension to part edges.

Dimensioning to Part Edges



Known Dimensions

Known dimensions allow you to establish meaningful parametric dependencies when creating a section of a feature.

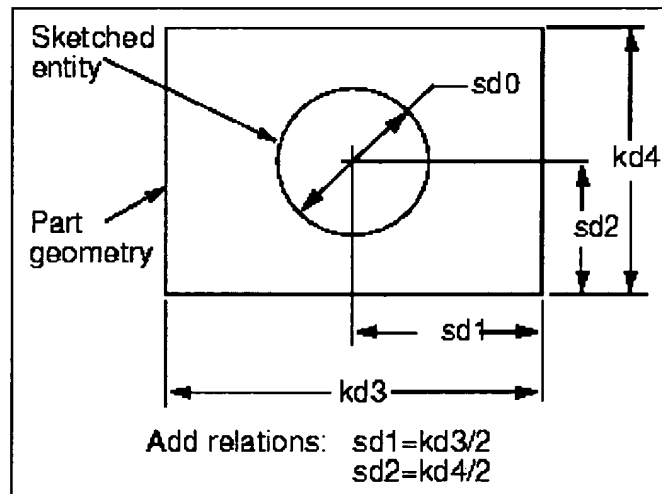
How to use known dimensions in Sketcher mode

1. Sketch and dimension as usual.
2. Create **Known** dimensions on part geometry that will be used to drive the feature section.
3. From the SKETCHER menu, choose **Relation**.

4. Add relations connecting **Normal** section dimensions with the **Known** ones (see the illustration Using Known and Normal Dimensions).
5. Regenerate. Values of **Normal** dimensions change according to the relations.

Dimensions driven by Sketcher relations cannot be modified directly. To access Sketcher relations, choose Redefine and Section, or choose Relations from the Part menu, Feat Rel, select the feature, and choose Section.

Using Known and Normal Dimensions



Using Automatic Dimensioning

With automatic dimensioning, the system adds dimensions to your sketch so it is fully constrained and then regenerates it. When you create a section in a part with existing geometry, the system must locate the section with respect to the part. For this, Sketcher uses "known" geometry. There are several ways to make geometry "known":

- When you specify horizontal and vertical references for orienting the sketching plane, the system uses these as known geometry.
- You can make geometry known by referencing it before you use AutoDim (for example, by aligning it, using a model edge for sketching, or dimensioning to it.)
- When you align the section to the part geometry at the system alignment query, that geometry becomes "known".

To make any known geometry unknown, use the Unalign command.

How to dimension a section automatically

1. Sketch geometry and choose **AutoDim**.
2. If the system needs additional references for locating the section with respect to the part geometry, it prompts you to select these references by picking edges and vertices. When finished, choose **Done**

Select from the GET SELECT menu or press the middle button.

3. The system adds all necessary dimensions to constrain the section. Notice that projections of known geometry onto the sketching plane appear in orange with a dotted font.
4. The system checks the section if it should be aligned to the part geometry. If such alignment is possible, the system brings up the query menu so you can select the desired action. Choose one of the following options:
 - **Align**-Align geometry as prompted in the alignment query.
 - **Don't Align**-Do not align geometry for that alignment query.
 - **Don't Align Any**-Do not align any geometry in the current auto dimensioning operation.
5. Once the system regenerates successfully, you can move dimensions to the desired location by using **Move** from the GEOM TOOLS menu and then **Dimension** from the MOVE ENTITY menu.

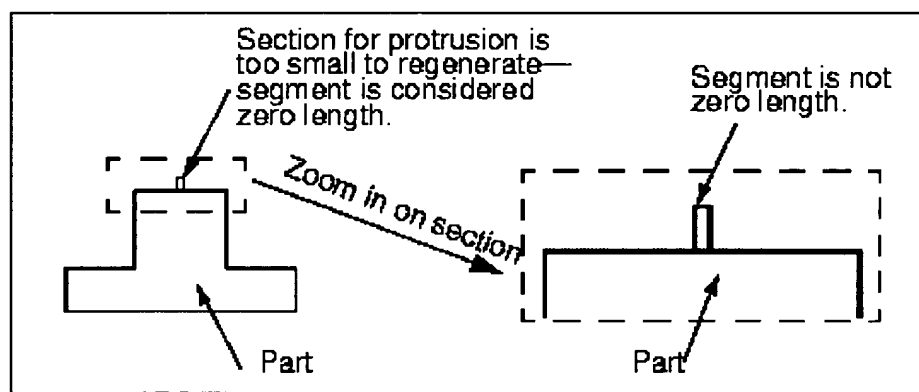
Note:

Any known geometry that is not used by the system is remembered by section and becomes its parent.

Regenerating a Section Sketch

During the section regeneration, Pro/ENGINEER checks the section geometry for validity and whether the section is placed adequately with respect to the part. If the section to be sketched is small, you should sketch in a sufficiently magnified view to avoid zero-length segments (see the illustration Magnified Sketching). Pro/ENGINEER regenerates the section with respect to the scale of the part on the screen when the section was sketched.

Magnified Sketching

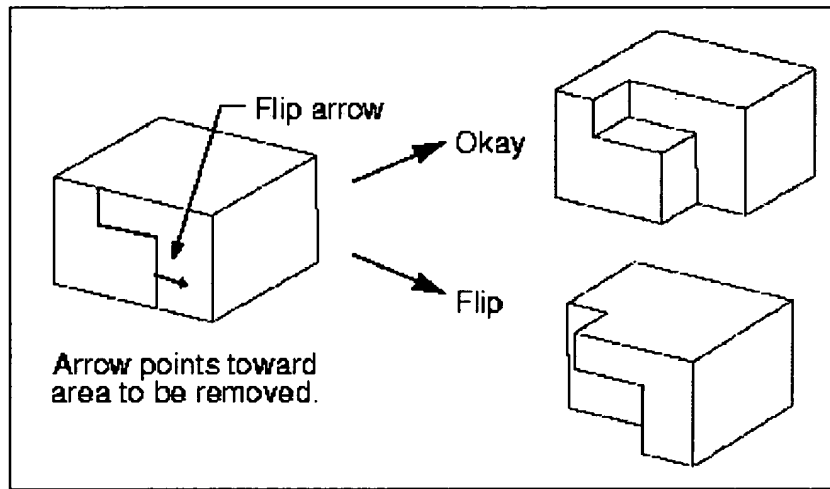


Another way to avoid zero-length segments is to sketch the section larger than actual size, then modify the dimension to the desired size. The dimensions can be relocated, if necessary, using the Sec Tools menu Move Dim option.

Specifying the Side

After the sketching is completed, the system displays a red arrow indicating the side where material is to be added or removed (see the illustration [Specifying the Side for a Cut Feature](#)). Use the Flip and Okay options in the Sel Side menu to specify the side.

Specifying the Side for a Cut Feature



Saving a Sketcher View

If you want to return to the current Sketcher view, save the view with a specific name before you exit Sketcher mode. To do this, choose View from the Main menu, Names from the Main View menu, Save from the Save/Retr menu, then enter a name for the view. To retrieve this view later, choose the command sequence View, Names, Retrieve and specify the view name.

Viewing the Results of the Sketch in 3D

To view the results of the completed sketch in 3D after exiting from Sketcher, choose View from the Main menu, Orientation from the Main View menu, and select a view (for example, Default or Spin) from the Orientation menu.

Feature Form Options

It is good practice to use the *simplest* feature possible. For example, if an extruded cut will work, do not use a general blend. Do not rule out any form for a feature. Each form option has its own advantages if used in the right place.

Specifying the Depth Element of a Feature

When you define the Depth element of a feature (for example, a protrusion, cut, slot, hole, or surface feature), the Spec To (or Spec From) menu appears with the following options:

- **Blind**-Enter a dimension for the feature depth. You can then control the feature depth by changing the depth dimension.

- **Thru Next**-Terminate the feature at the next part surface.
- **Thru All**-The new feature intersects all surfaces.
- **Thru Until**-Extend the feature until the intersection with the specified surface.
- **UpTo Pnt/Vtx**-Specify the depth up to a plane parallel to the sketching plane, and passing through the selected datum point or vertex.
- **UpTo Curve**-Specify the depth up to a plane parallel to the sketching plane, and passing through the selected edge, axis, or datum curve.
- **UpTo Surface**-Specify the depth up to a selected surface.

Blind

A blind feature has an independent parameter that governs its depth. In the special case of a sketched blind hole, the depth is indicated in the sketched cross-section. With other features, Pro/ENGINEER prompts you for a depth. Enter a value, or enter <CR> to accept the default value. The system extrudes the feature to the specified depth.

"Through" Options

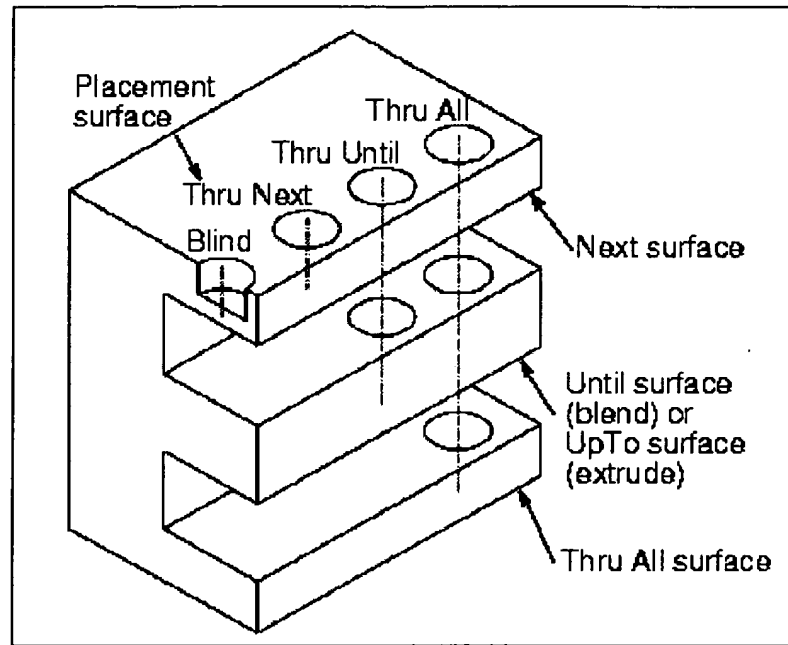
The system creates a "through" feature from the placement surface until its intersection with the specified termination surface.

Consider the following rules for using the through options:

- For all through intersections, the feature being created must lie entirely within the surface (or quilt) on which it is terminated.
- When you use the Thru Until option for an extruded feature, the feature cannot terminate on a datum plane. Use the UpTo Surface option to select the terminating datum.
- When you use the Thru Until option for a blend, you can select a datum plane to terminate the feature, but the datum plane must be parallel to the sketching plane.
- Protrusions created with the Thru Next option cannot terminate on a datum plane.
- Thru Next, Thru Until, and Thru All are not available when you create surface features.
- Thru All is available for protrusions only if the part has existing geometry.

The illustration Depth Options for Removing Material shows the valid depth options for removing material in holes, cuts, and slots.

Depth Options for Removing Material

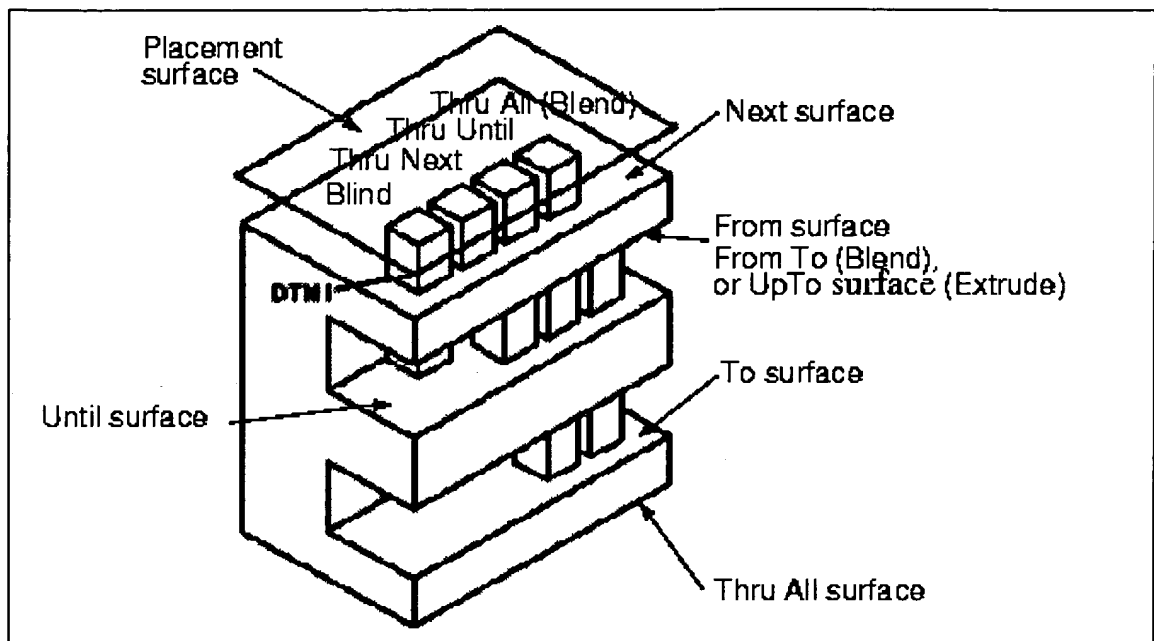


Some tips for using the "through" options:

- Use a through option (Thru All, Thru Until, or Thru Next) when you want the feature to terminate on a specified surface.
- Use Thru Next when the feature should stop at the first surface it reaches.
- Use Thru All when the feature should stop at the last surface it reaches.
- Use Thru Until when you want to pick the termination surface.
- Through features do not have a parameter associated with the extrusion depth. Modifying the terminating surface alters the depth of the feature.

The following illustration shows valid depth options for adding material.

Depth Options for Adding Material



The "Up To" Options

The "up to" options are available for extruded (protrusions, cuts, and slots), revolved, and surface features.

When you use UpTo Surface option, you can select an existing surface or create a datum plane. Choose the desired method by selecting one of the following options:

- **Select Surf**-Select any part surface, quilt (composed of one or more surfaces), or datum plane.
- **Make Datum**-Create a datum plane be used as an Up To reference.

For solid features, you can select the surfaces of the following types:

- Another part surface, which need not be planar (see the illustration [Using Both Sides and Up To Surface](#))
- A datum plane, which need not be parallel to the sketching plane
- Quilt composed of one or more surfaces

Note:

For a surface feature, the terminating surface can only be a datum plane, which must be parallel to the sketching plane.

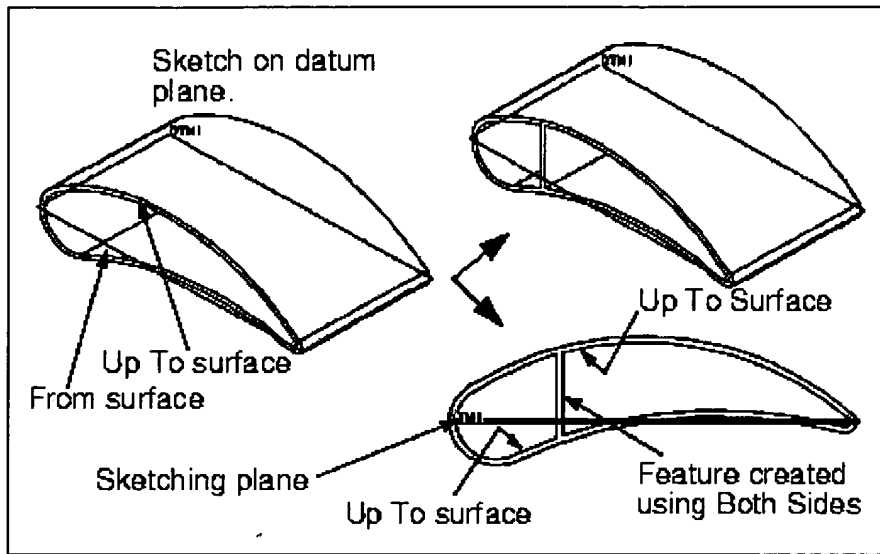
When creating features in Assembly mode, you can select geometry of another component as references for the "up to" options.

Using a quilt as the terminating surface allows you to create features intersecting with multiple surfaces and is very useful for creating patterns consisting of multiple terminating surfaces.

The illustration [Using Both Sides and Up To Surface](#) shows an example of using a datum plane as an "up

to" reference.

Using Both Sides and Up To Surface



The "From To" Option

The From To option is applicable for blends only (for extrusions and surface features, use Both Sides and Up To Surface). The From To option extrudes a feature from a selected surface to another surface. It is designed to create features between sculptured surfaces, but can be used for any type of surface, with the following restrictions:

- Intersection surfaces must be physical surfaces, therefore datum planes are not allowed as "From" or "To" surfaces.
- The feature section must intersect the From To surfaces completely.

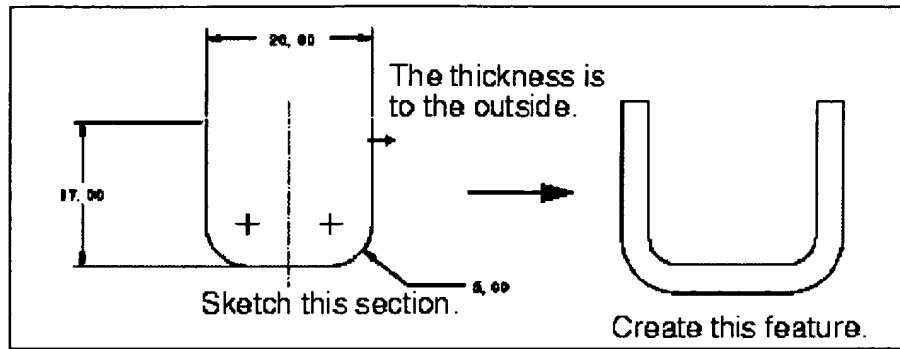
Note:

The From To option is not a replacement for through options. Use the proper "through" options whenever possible.

Thin Features

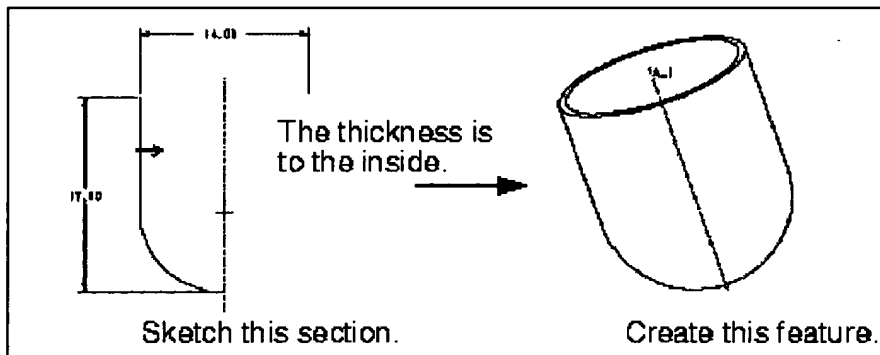
By default, any of the options in the Solid Opts menu will be created using Solid, that is, with no thickness. The Pro/FEATURE Solid Opts menu Thin option creates simplified section sketches with a uniform thickness (see the illustration Extruded Thin Feature) by applying a thickness to the section as it is extruded, revolved, swept, imported, or blended (for blends that are general, rotational, or closed).

Extruded Thin Feature



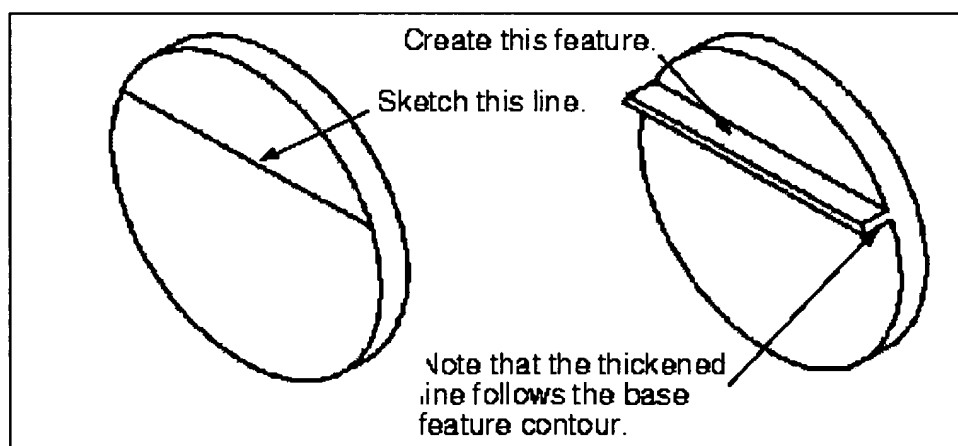
Thin features can be used as a base feature, or as cuts, slots, or protrusions in secondary features. You can add material to a thin feature to either side of the sketched section entities using the Thin Opt menu option Flip or Okay (to specify the side) or Both (equally to both sides of the section entities). You can modify both the section and thickness after the feature is created.

Revolved Thin Feature



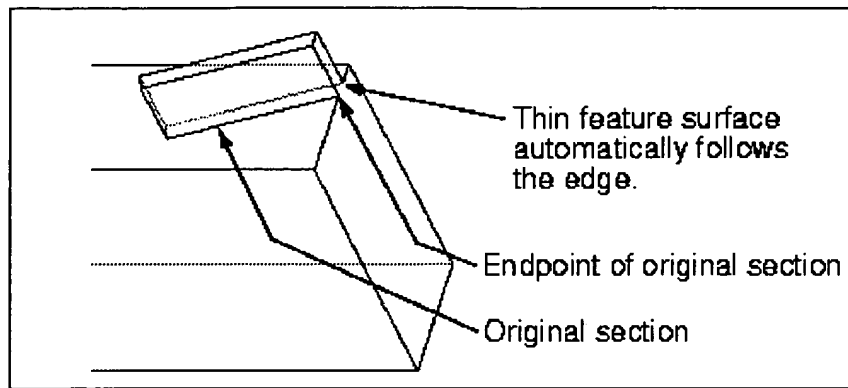
Thin features follow the contour of the part if an endpoint of the section is on a part edge (see the illustration [Terminating Thin Feature Edges](#)).

Terminating Thin Feature Edges



If the endpoint is on a single edge only, the corresponding created surface will follow that edge (see the illustration [Thin Feature Endpoint on a Single Edge](#)).

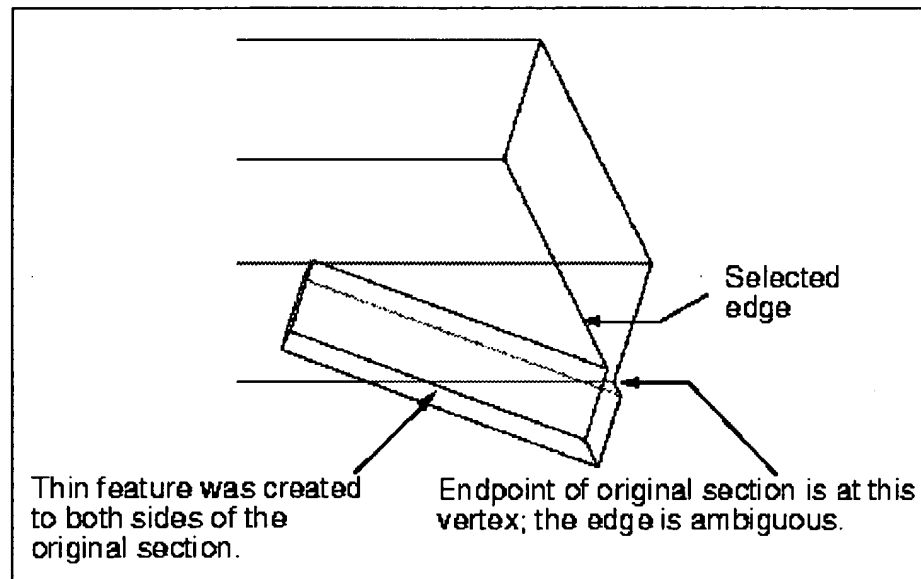
Thin Feature Endpoint on a Single Edge



If a thin section endpoint is located at a vertex, it lies on multiple edges. In this case, Pro/ENGINEER prompts you (with a small red circle and crosshair at the section endpoint) to select the edge that the end surface will follow (see the illustration [Thin Feature Endpoint on a Vertex](#)).

You cannot include text in a sketch of a thin feature (see [Using Sketcher Text](#)).

Thin Feature Endpoint on a Vertex



How to create thin features

1. Choose **Protrusion**, **Cut**, or **Slot** from the SOLID menu.
2. Choose **Thin** and **Done** from the SOLID OPTS menu.
3. Choose the extrusion attribute (see [Extrusion Attributes](#)).
4. Select a sketching plane and orient the section.
5. Sketch the feature section. Remember that thickness is being added automatically, so you can use a simplified "stick figure" approach to the sketch.

6. Choose a thin feature direction using the THIN OPT menu (see [Thin Features](#)).
7. Enter the thickness of the thin section.
8. Choose a depth option and enter a depth, if required. See [Specifying the Depth Element of a Feature](#) for more information.
9. If an endpoint of the sketched feature terminates on a part vertex that causes ambiguity, you must select the terminating edge or surface for the highlighted endpoint of the sketched section. Selecting **Done Sel** without an edge or surface causes the thin feature to be terminated with its end face normal to the sketched section-it will not follow the part contour.

Extrude

The Extrude option creates a feature that is formed by projecting the section straight away from the sketching plane (see the illustration [Sketching Extruded Features](#)). It is the most basic and frequently-used form option.

Extrusion Attributes

The depth of an extruded feature (except of the base solid feature, which is Blind by default) must be specified by choosing a direction attribute, then the desired depth options. See [Specifying the Depth Element of a Feature](#) for a detailed explanation of valid depth options and termination surfaces.

The direction attributes specify the location of the extruded feature with respect to the sketching plane. The possible values are as follows:

- **One Side**-Specify the depth of the feature to one side of the sketching plane.
- **Both Sides**-Specify depth for both sides of the sketching plane separately.

Using One Side and Both Sides with Depth Options

If you choose One Side, the feature starts from the sketching plane and is extruded in the direction of feature creation according to the selected depth option.

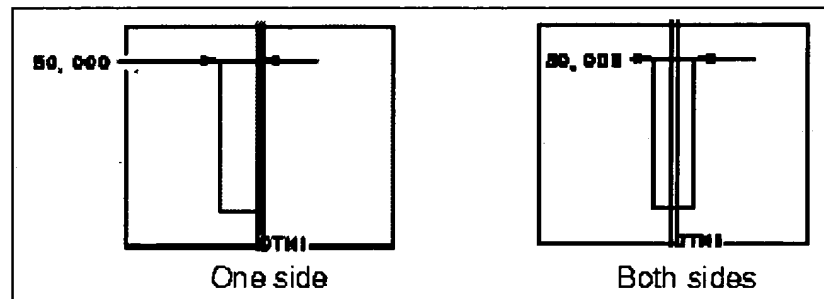
If you choose Both Sides, for options other than Blind, you must define the from and to sides of the feature. Which side is considered from or to depends on the direction of feature creation.

Note the following rules for the "both sides" features:

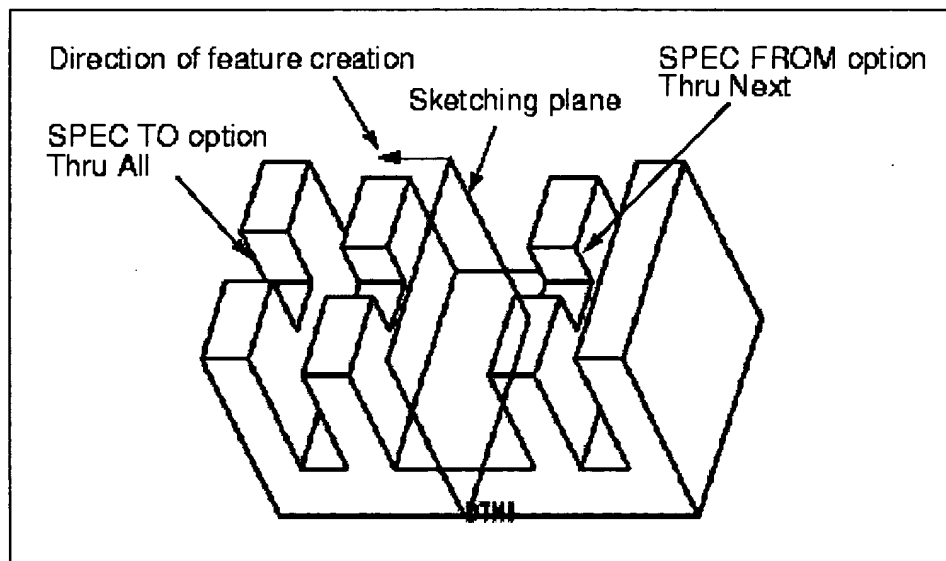
- For a Blind feature, the depth you enter is divided symmetrically by the sketching plane (see the illustration [Location of a Blind Feature Relative to the Sketching Plane on page 4 - 37](#)).
- The system applies all through options with respect to the sketching plane. For example, if you select Thru Next from the Spec From menu, the system looks for the next valid surface from the sketching plane, in the direction opposite to the direction of feature creation (see the illustration [Thru Options for Both Sides](#)).

- Up to options allow you to locate the feature completely aside from the sketching plane (see the illustration Example of Using Up To). They also allow you to use vertices, edges, datum planes, and non-planar surfaces as termination references.

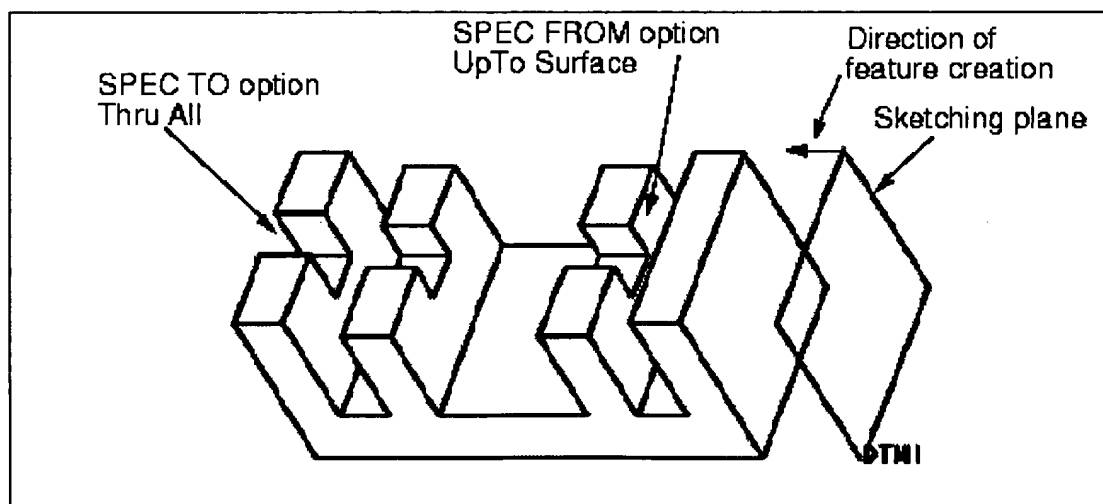
Location of a Blind Feature Relative to the Sketching Plane



Thru Options for Both Sides



Example of Using Up To



Sketching an Extruded Feature Section

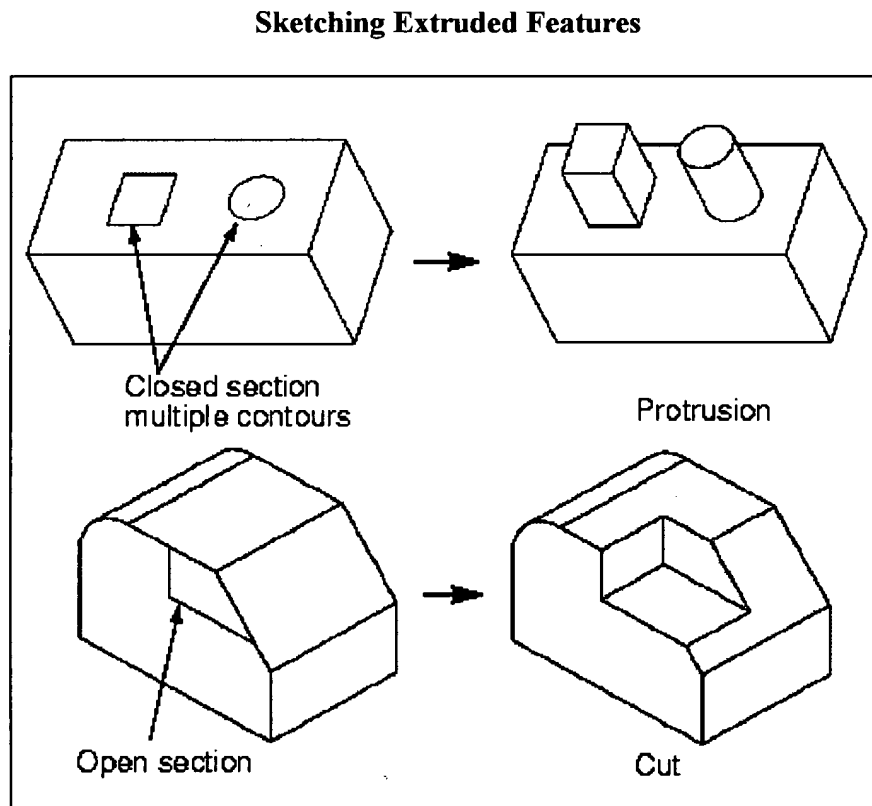
Extruded sections can be opened or closed. Note the following:

- Open sections cannot have more than one loop. All open ends should be explicitly aligned to the part edges.
- Closed sections may consist of one or more closed outside loops, or of one outside loop with one or more inside loops. In the last case, Pro/ENGINEER takes the largest loop as the outside, and each other loop is considered to be a hole in the large loop. The loops must *not* intersect each other.

Specifying the Direction of an Extruded Feature

For some protrusions and most cuts, the system displays a red arrow after the sketch is completed, showing the side where material will be added or removed. Use the Direction menu options Flip and Okay to specify the side.

The following figure illustrates some sketched extruded features.



How to create an extruded feature

1. Choose **Extrude** and **Done** from the SOLID OPTS menu.
2. Choose **One Side** or **Both Sides** and **Done** from the ATTRIBUTES menu.
3. Select or create the sketching plane. Specify the direction of feature creation using **Flip** and **Okay**.

- Select or create the sketching reference.
4. Sketch and regenerate the section, then choose **Done**.
 5. If prompted, specify the side to add or remove material using **Flip** and **Okay**.
 6. Choose a depth option. If you chose **One Side**, choose an option from the SPEC TO menu, then **Done**. If you choose **Thru Until** or an "Up To" option, select the termination reference (surface, edge, and so on). See [Specifying the Depth Element of a Feature](#) for more information on depth options.

...or...

If you chose **Both Sides**, choose an option from the SPEC FROM menu, then **Done**. Select a termination reference or enter a depth if needed. Unless you chose **Blind**, the system displays the SPEC TO menu. Choose the depth option for the second side, then **Done**. Select the termination reference, if needed.

7. If you chose **Blind**, enter a value for the extrusion depth.

Changing Extruded Features

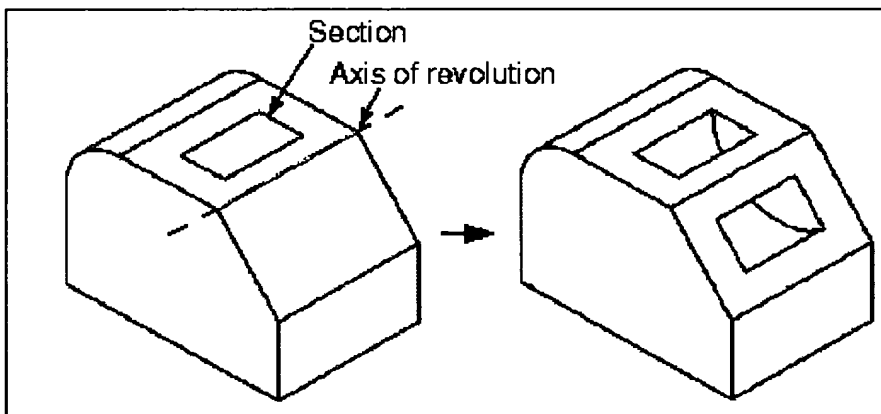
At any time, you can modify the dimension values of an extruded feature using the Modify option. Using the Redefine option, you can redefine any feature elements. For more information on redefining elements, see the section [Redefining Features Redefining Features](#).

You can also change the sketching plane and depth references using the Reroute option (see [Rerouting Features](#)).

Revolve

The Revolve option creates a feature by revolving the sketched section around a centerline from the sketching plane into the part (see the illustration [Revolve Cut or Slot](#)). When sketching the feature, the first centerline sketched is the axis of revolution. The section must lie completely on one side of this centerline and must be closed.

Revolve Cut or Slot



A revolved feature can be created either entirely on one side of the sketching plane, or symmetrically on both sides of the sketching plane.

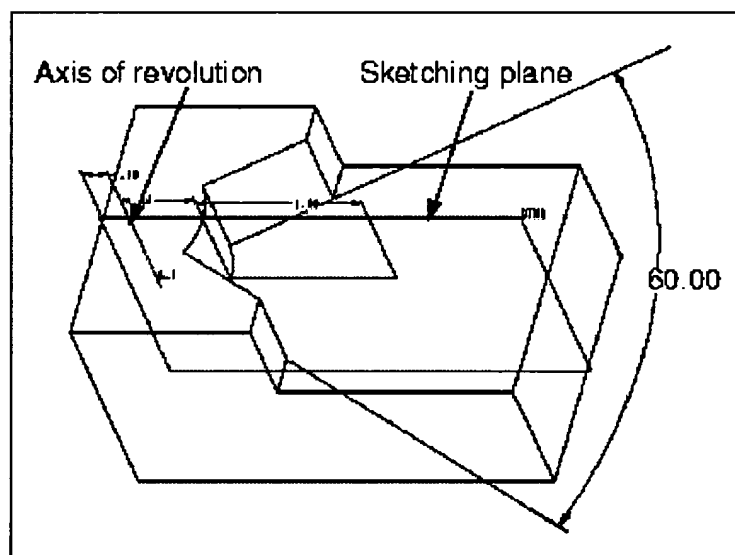
To create or redefine a Revolve feature, specify the elements in the following order:

- Attributes
- Section
- Direction
- Angle

Specifying the Revolved Feature Attributes

The Attributes menu elements One Side and Both Sides are available for all but the first feature. For Both Sides (see the illustration Revolved Cut or Slot-Both Sides Option), the feature will be revolved symmetrically in each direction for one half of the angle specified in the Options menu, whether preset or variable.

Revolved Cut or Slot-Both Sides Option



Sketching the Revolved Feature Section

To create a revolved section, create a centerline and then sketch the geometry that will be revolved about that centerline.

Notes:

- The revolved section must have a centerline.
- The geometry must be sketched on only one side of the centerline. If geometry is sketched on

both sides, Pro/ENGINEER issues a warning and remains in Sketcher mode.

- If you use more than one centerline in the sketch, Pro/ENGINEER uses the first centerline sketched as the axis of rotation.

Specifying the Angle of Revolution

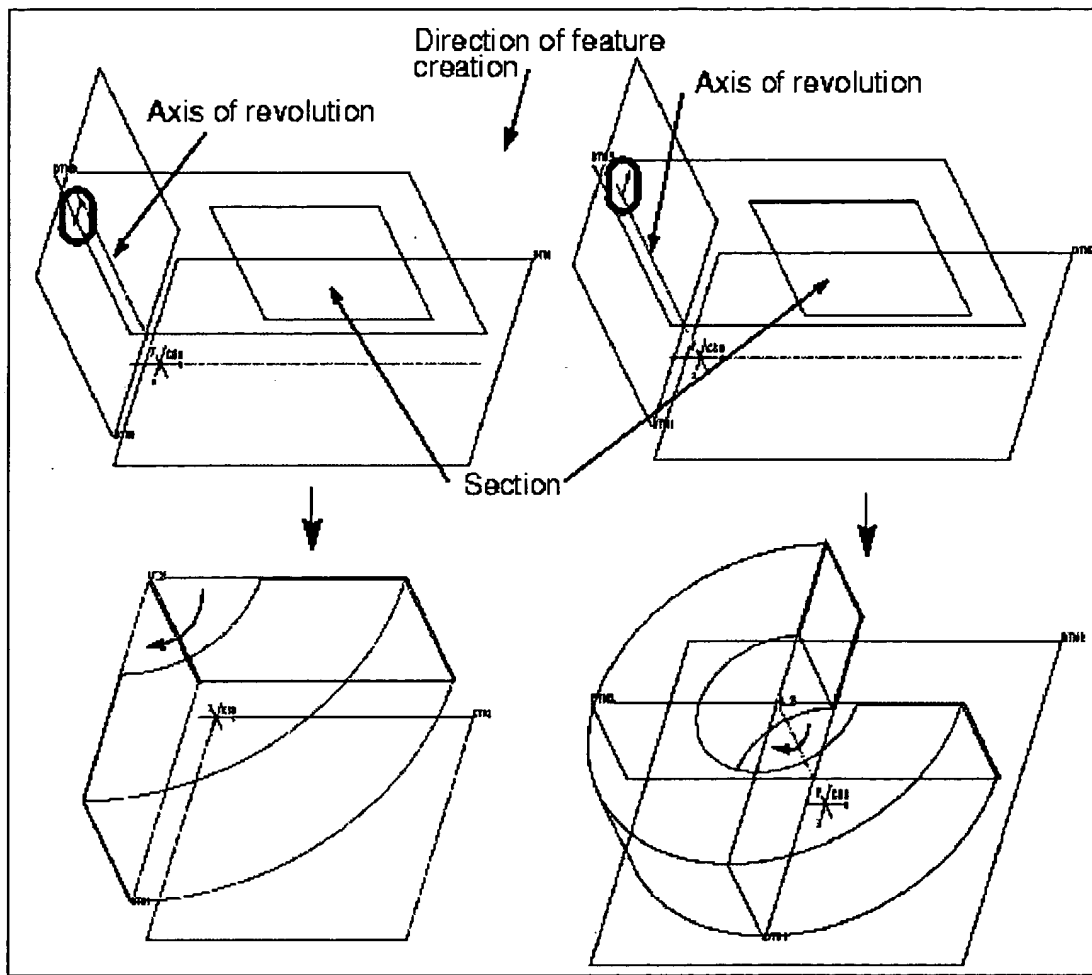
After choosing Revolve, the system displays the Rev To menu. This menu allows you to specify the angle of revolution of the feature, and whether that angle is to be measured entirely on one side of the sketching plane, or symmetrically on both sides of the sketching plane.

The Rev To menu options are as follows:

- **Variable**-Specify any angle of revolution less than 360 degrees. The angle is controlled by a dimension that the system displays when you modify the part, and in drawings. A corresponding dimension will not appear if you choose a preset angle.
- **90**-Create the feature with a fixed angle of 90 degrees.
- **180**-Create the feature with a fixed angle of 180 degrees.
- **270**-Create the feature with a fixed angle of 270 degrees.
- **360**-Create the feature with a fixed angle of 360 degrees.
- **UpTo Pnt/Vtx**-Create the revolved feature up to a point or vertex. The revolved feature ends when the section plane reaches the point or vertex (see the illustration Revolve Feature Up To Option Selections).
- **UpTo Plane**-Create the revolved feature up to an existing plane or planar surface that must contain the axis of revolution. If you are revolving to a datum plane, identify the plane and use the flip arrow to indicate on which side of the axis of revolution to stop revolving when the feature reaches the datum plane (as it is created in the direction of revolution). The feature ends when its revolving section plane reaches the plane (see the illustration Creating a Revolve Feature with UpTo Plane). If you are revolving to a non-datum plane surface, indicate the side by the location at which you select the surface as a reference (see the illustration Revolve Feature Up To Option Selections).

In the example in the illustration Creating a Revolve Feature with UpTo Plane, the revolve feature is created using the UpTo Plane option by selecting DTM1. Depending on the direction of the axis of revolution, you will get two different results.

Creating a Revolve Feature with UpTo Plane



Changing Revolved Features

At any time, you can modify the dimension values of a revolved feature using the Modify option. Using the Redefine option, you can redefine the elements (for more information, see [Redefining Features](#)).

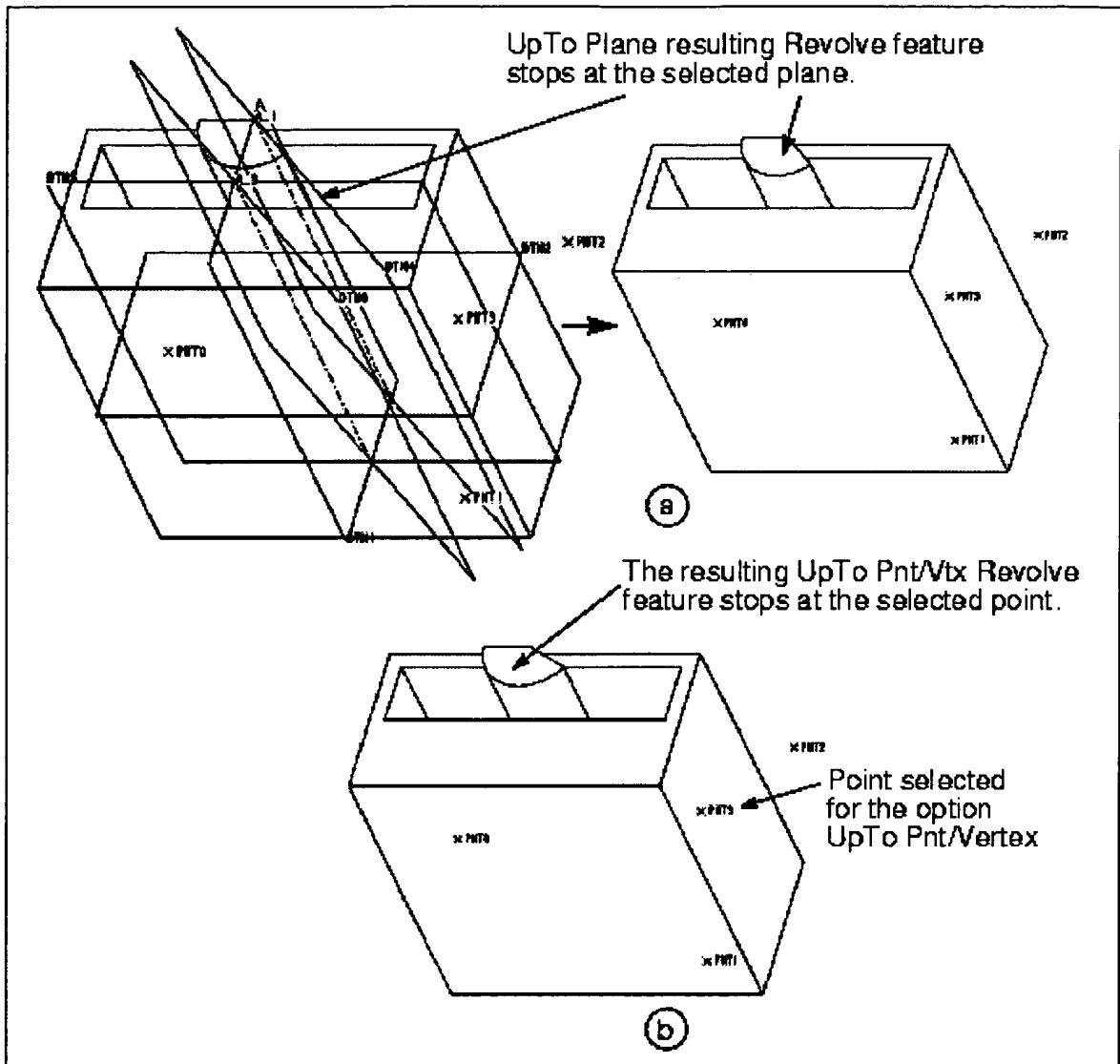
You can redefine the following elements:

- Attributes
- Section
- Material side (for a Thin feature)
- Direction
- Angle

You can also change the sketching plane and depth references using the Reroute option (see [Rerouting Features](#)).

[Revolve Feature Up To Option Selections](#) illustrates a revolved feature that uses the Up To options.

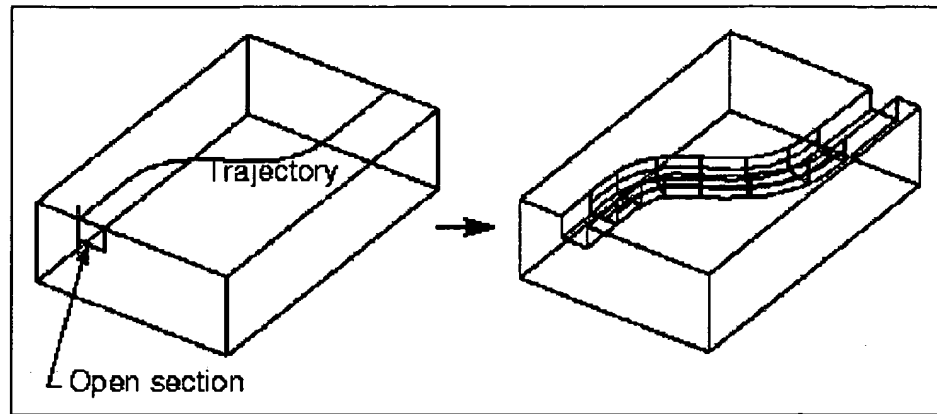
Revolve Feature Up To Option Selections



Sweep

A sweep is created by sketching or selecting a trajectory, then sketching a section to follow along it. You can create more advanced sweeps using the Advanced option (see [Advanced Form Features](#)).

Swept Cut



Rules for Defining a Trajectory

A constant section sweep can use either a trajectory sketched at the time of feature creation, or a trajectory made up of selected datum curves or edges. As a general rule, the trajectory must have adjacent reference surfaces, or be planar. When you define a sweep, the system checks the specified trajectory for validity and establishes normal surfaces. When ambiguity exists, the system prompts you to select a normal surface.

Depending on the type of chain selected as a trajectory, the system behaves as follows:

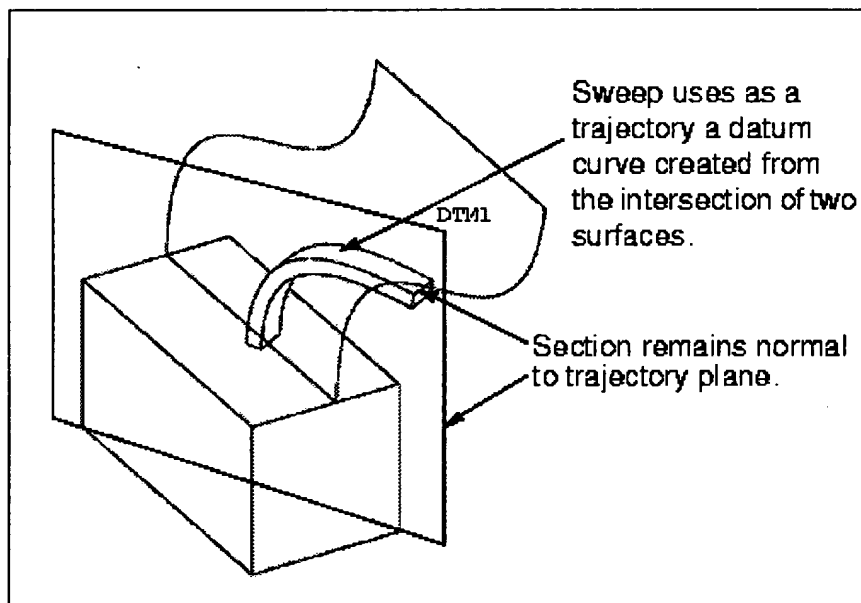
- All chain segments reference edges-The normal surfaces are the adjacent surfaces of the edges. If the edges are two-sided, the system prompts you to choose one set of surfaces.
- All chain segments reference entities that belong to a datum curve, created by referencing surfaces (for example, by using the Projected option)-The normal surfaces are reference surfaces of the curve. If the curve references two sets of surfaces, the system prompts you to choose one.
- All chain segments reference a sketched datum curve-the normal surface is the sketching plane of the curve.
- The chain of edges/curves is planar (other than a straight line)-The normal surface is the plane defined by the chain.
- Datum curves that you select for the trajectory must be created with one of the following options:
 - **Sketch**
 - **Intr. Surfs**
 - **Use Xsec**
 - **Projected**
 - **Formed**
 - **Offset In Srf**
 - **2 Projections**

- **From Equation** (only for curves that are in a plane)

Consider the following special cases:

- If a datum curve and its adjacent surfaces were bent by a toroidal bend feature, you can use that curve as a trajectory.
- For a variable section sweep, any chain of edges/curves is valid, and establishing normal surfaces is not required.
- If you extend the chain with Trim/Extend in the Chain menu, the system accepts that chain if it is planar.

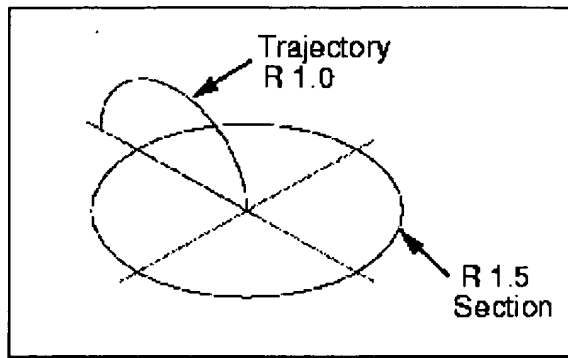
The following figure illustrates a constant section sweep.



Note that sweep trajectories cause the creation of a feature to fail if the feature intersects itself because of the following:

- A trajectory crosses itself.
- An arc or a spline radius is too small, relative to the section, and the feature intersects itself traversing around the arc (see the following illustration).

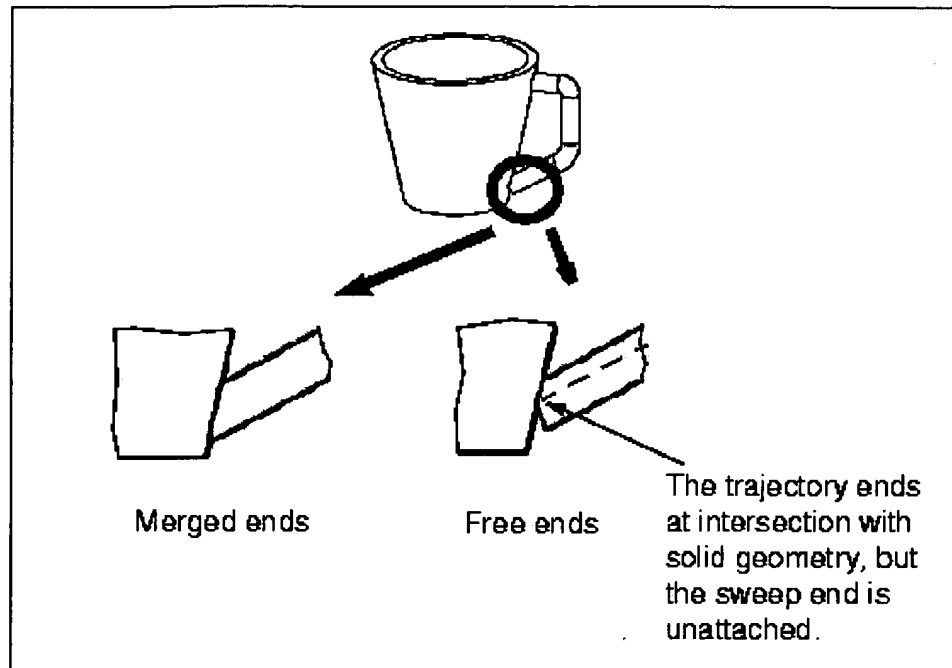
Self-Intersecting Feature



How to create a swept feature

1. Use the command sequence **Feature, Create, Solid, Protrusion**.
2. Choose **Sweep** and **Done** from the SOLID OPTS menu.
3. Pro/ENGINEER displays the feature creation dialog box for sweeps.
4. Sketch or select the trajectory using a SWEEP TRAJ menu option. The trajectory can be open or closed. The options are as follows:
 - **Sketch Traj**-Sketch the sweep trajectory using Sketcher mode.
 - **Select Traj**-Select a chain of existing curves or edges as the sweep trajectory. The CHAIN menu allows you to select the desired trajectory (see [Chain Processing](#)).
5. If the trajectory lies in more than one surface, such as a trajectory defined by a datum curve created using **Intr. Surfs**, the system prompts you to select a normal surface for the sweep cross-section. Pro/ENGINEER orients the Y-axis of the cross-section to be normal to this surface along the trajectory.
6. Create or retrieve the section to be swept along the trajectory and dimension it relative to the crosshairs displayed on the trajectory. Choose **Done**.
7. If the trajectory is open (the start and end points of the trajectory do not touch (see the illustration [Free and Merged Ends](#))) and you are creating a solid sweep, choose an ATTRIBUTES menu option, then **Done**. The possible options are as follows:
 - **Merge Ends**-Merge the ends of the sweep, if possible, into the adjacent solid. To do this, the sweep endpoint must be attached to part geometry.
 - **Free Ends**-Do not attach the sweep end to adjacent geometry.

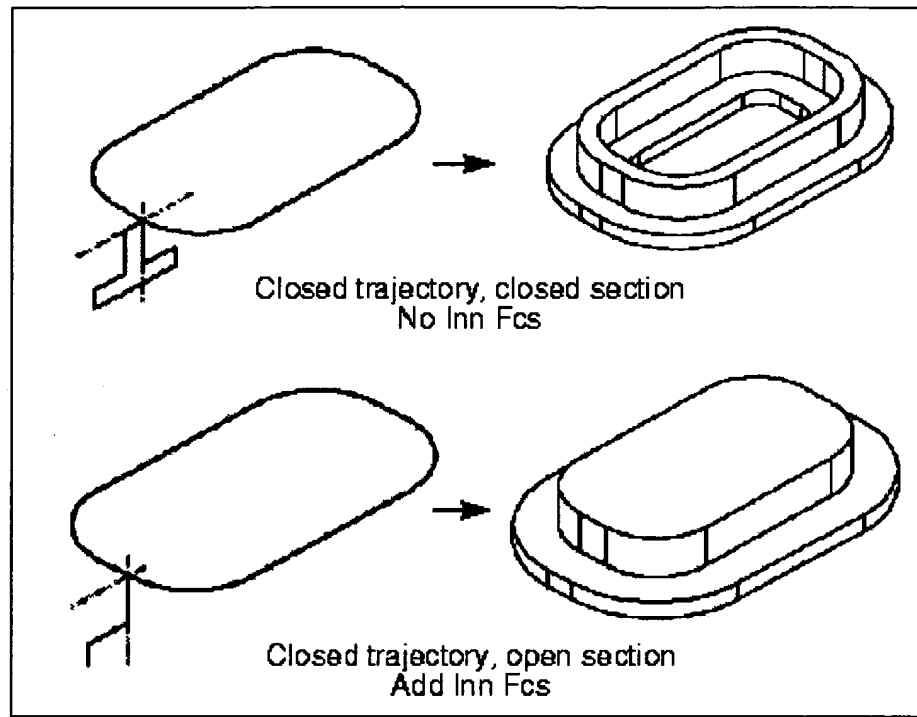
Free and Merged Ends



8. If the sweep trajectory is closed (see the illustration Different Sweep Trajectories and Sections), choose one of the following SWEEP OPT menu options and **Done**:
 - **Add Inn Fcs**-For open sections, add top and bottom faces to close the swept solid (planar, closed trajectory, and open section). The resulting feature consists of surfaces created by sweeping the section and has two planar surfaces that cap the open ends.
 - **No Inn Fcs**-Do not add top and bottom faces.
9. Choose **Flip**, if desired, then **Okay** from the DIRECTION menu to select the side on which to remove material for swept cuts.
10. The system issues a message stating that all the elements have been defined. If desired, select one of the buttons in the dialog box.
11. Select **OK** in the dialog box to create the sweep.

You can redefine sweep sections or trajectories after the feature is created. To do this, choose **Redefine**, select the sweep, and Pro/ENGINEER displays the feature creation dialog box. You can redefine the Trajectory or Section elements.

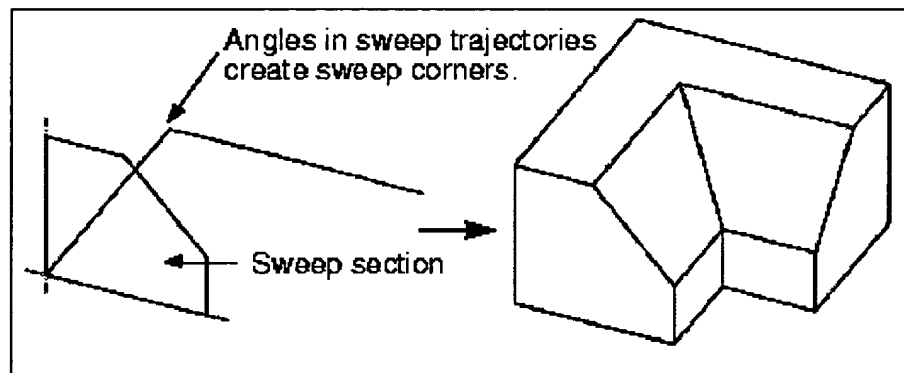
Different Sweep Trajectories and Sections



Swept Feature Corners

A trajectory can be sketched with two straight line segments that form an angle. The resulting sweep will have a mitered corner (see the following illustration).

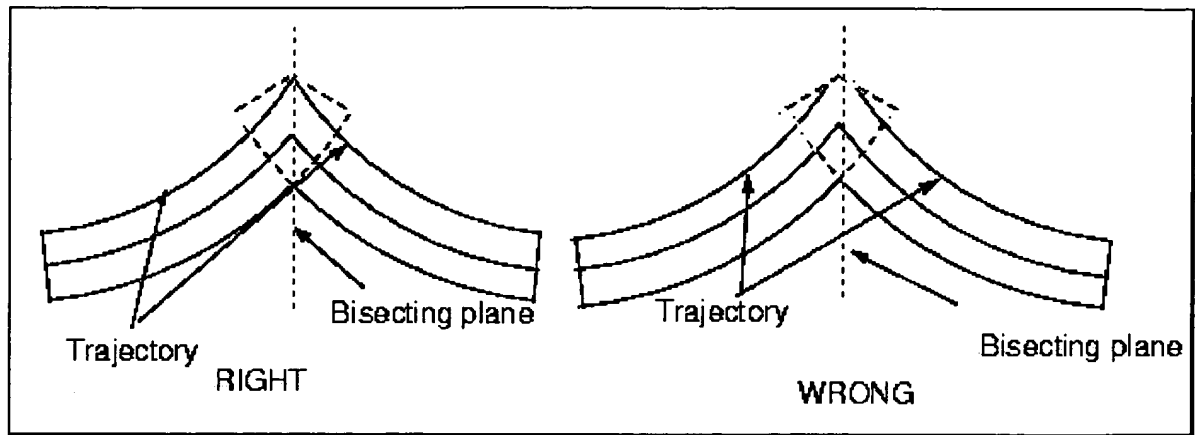
Sweep with Mitered Corner



Non-Tangent Trajectory Segments

Sweeps can be made along trajectories consisting of non-tangent entities. However, as it is swept along extruded non-tangent entities, the sweep section geometry must completely cross a plane that bisects the angle between two trajectory entities (as shown in the following illustration).

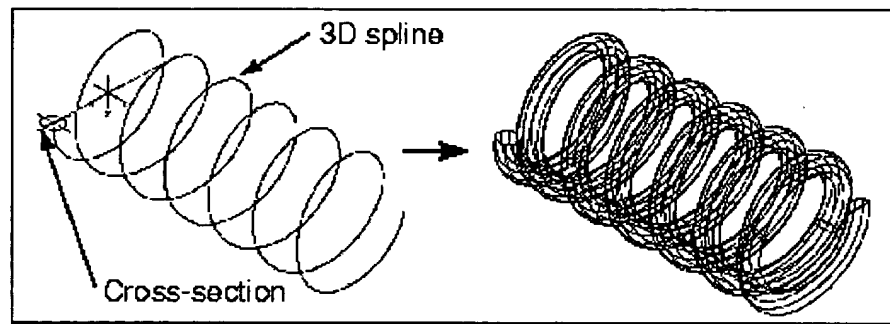
Sweeping Along Non-Tangent Entities



Three-Dimensional Sweeps

With Pro/FEATURE, sweeps can be created along a three-dimensional path by creating a three-dimensional spline for the sweep trajectory. That is, Pro/FEATURE allows you to modify the Z-coordinates of spline points (all other Sketcher entities must lie on a two-dimensional sketching plane). In all other respects, three-dimensional sweeps are created the same as two-dimensional sweeps. Beyond the three-dimensional sweeps described here, for such applications as creating springs, you can create an advanced feature helical sweep by sweeping a section along a helical trajectory (see the illustration [Sample Area Graph and Information Window](#)).

Spring Created from a 3D Spline



How to create a three-dimensional spline

1. Create a two-dimensional spline and dimension it to a Sketcher coordinate system.
2. Modify the X-, Y-, and Z-coordinates for one or more spline points. You can modify the spline coordinates manually, or by using a spline definition file. See [Sketcher](#), for more information on modifying splines.

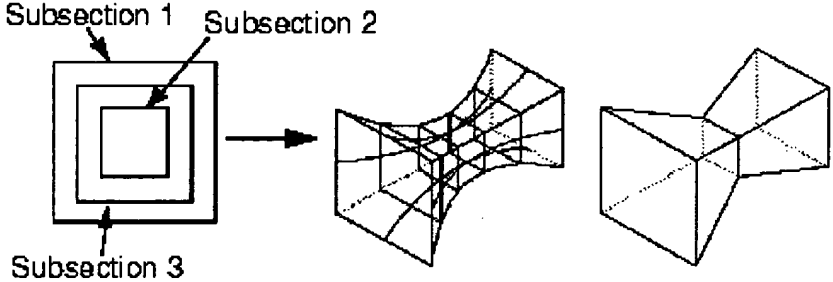
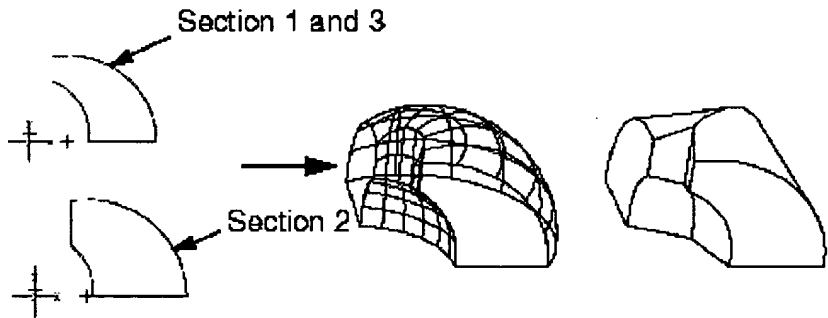
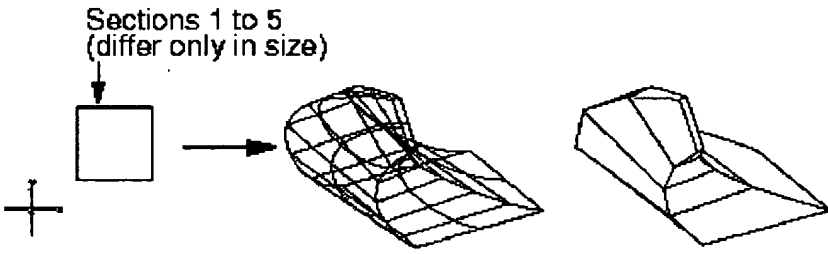
If the endpoints of the spline are attached to other entities in a sketch, Pro/ENGINEER ignores any changes to their coordinates.

Blend

A blended feature consists of a series of at least two planar sections that Pro/ENGINEER joins together at

their edges with transitional surfaces to form a continuous feature. Blends with parallel sections can be created in basic Pro/ENGINEER, but the Pro/FEATURE and Pro/SURFACE modules are required to create blends from non-parallel sections. See the table on for a complete breakdown of blend functionality by module.

Blend Types

Blend Type	Section -Smooth -Straight
<p>Parallel-All blend sections lie on parallel planes in one section sketch.</p>	
<p>Rotational-Blend sections are rotated about the Y-axis, up to a maximum of 120 degrees. Each section is sketched individually and aligned using the coordinate system of the section.</p>	
<p>General-Sections of a general blend can be rotated about and translated along the X-, Y-, and Z-axes. Each section is sketched individually, and aligned using the coordinate system of the section.</p>	

Techniques Common to All Blend Types

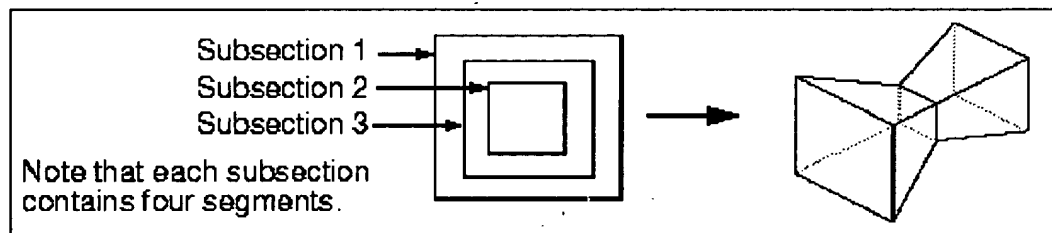
This section describes the techniques that are common to all blend types. The topics are as follows:

- Blend sections
- Starting point of a section
- Smooth and straight attributes

Blend Sections

The illustration Straight Parallel Blend shows a parallel blend for which the section consists of three subsections. Each segment in the subsection is matched with a segment in the following subsection; the blended surfaces are created between the corresponding segments. With the exception of capping a blend (see Capping Blends), blends must always have the same number of entities in each section. It is possible to make surfaces of non-parallel blends and parallel smooth blends disappear using a Blend Vertex (see 4 - 61).

Straight Parallel Blend

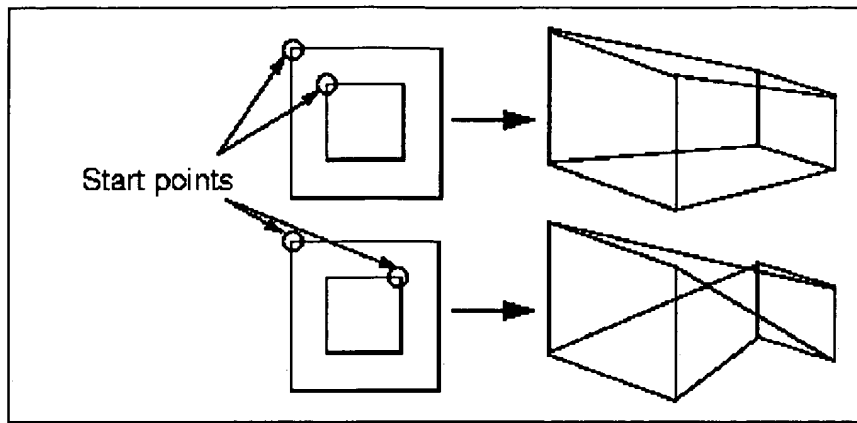


Starting Point of a Section

To create the transitional surfaces, Pro/ENGINEER connects the starting points of the sections and continues to connect the vertices of the sections in a clockwise manner. By changing the starting point of a blend section, you can create blended surfaces that twist between the sections (see the illustration Starting Points and Blend Shape).

The default starting point is the first point sketched in the subsection. You can place the starting point at the endpoint of another segment by choosing the option Start Point from the Sec Tools menu and selecting the point.

Starting Points and Blend Shape

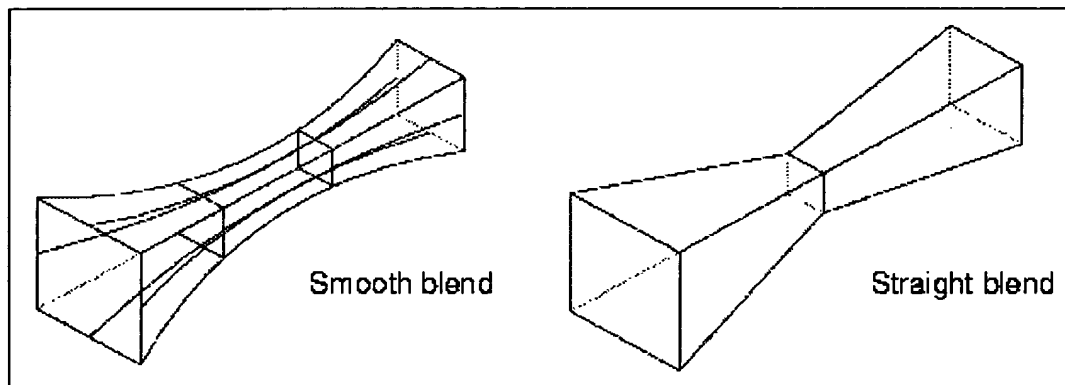


Smooth and Straight Attributes

Blends use one of the following transitional surface Attributes menu options:

- **Straight**-Create a straight blend by connecting vertices of different subsections with straight lines. Edges of the sections are connected with ruled surfaces.
- **Smooth**-Create a smooth blend by connecting vertices of different subsections with smooth curves. Edges of the sections are connected with spline surfaces.

Smooth and Straight Blends



How to create a blend

1. Use the command sequence **Feature, Create, Solid, Protrusion**.
2. Choose **Blend** and **Solid** or **Thin** from the SOLID OPTS menu, then **Done**.
3. Choose options from the BLEND OPTS menu, then **Done**.

The BLEND OPTS menu options are as follows:

- **Parallel**-All blend sections lie on parallel planes in one section sketch. For more information, see [Parallel Blends](#).
- **Rotational**-The blend sections are rotated about the Y-axis, up to a maximum of 120°. Each

section is sketched individually and aligned using the coordinate system of the section. For Rotational and General blends, any section coordinate systems are created in Sketcher mode. You cannot use the default coordinate systems. For more information, see [Rotational Blends](#).

- **General**-The sections of a general blend can be rotated about and translated along the X-, Y-, and Z-axes. Each section is sketched individually and aligned using the coordinate system of the section. For more information, see [General Blend](#).
- **Regular Sec**-The feature will use the regular sketching plane.
- **Project Sec**-The feature will use the projection of the section on the selected surface. This is used for parallel blends only. For more information, see [Projected Parallel Blend](#).
- **Select Sec**-Select section entities. This option is not available for parallel blends.
- **Sketch Sec**-Sketch section entities.

Parallel Blends

You create parallel blends using the Parallel option in the Blend Opts menu. A parallel blend is created from a single section that contains multiple sketches, called subsections. First and last subsections can be defined as a point or a blend vertex.

Whenever you modify or redefine the section for a parallel blend feature, the system displays the dimensions and contours for all the subsections.

Note that if you make cuts in a parallel projected blend, the sections *must* be closed.

Parallel Blend Sections

Parallel blend sections do not behave like ordinary sections. A parallel blend section cannot be retrieved into Sketcher mode or into any feature other than a parallel blend. You can retrieve a saved section using Place Section (see [Sketcher Grid](#)) only when the blend is a secondary feature and is going to be placed on an existing feature. The retrieved section will be added to the current subsection and can be placed into different subsections with variations in rotation angle and size.

How to save a parallel blend section

1. Save the section while you are creating the parallel blend feature in Sketcher mode.
2. At any other time in the session where the section was created, choose **Save** from the DBMS menu and enter the name of the section.

How to retrieve a parallel blend section

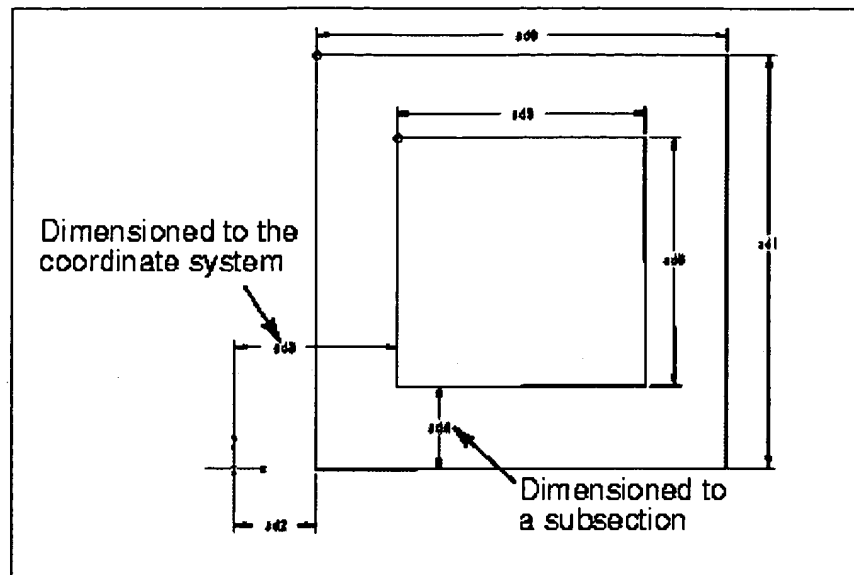
1. Begin to create a parallel blend as a secondary feature. Choose a sketching plane and a reference plane on an existing feature.
2. When you choose **Sec Tools**, the system displays the SEC TOOLS menu. Choose **Place Section**.

3. Enter or choose the section name.
4. The system displays the section in a subwindow.

How to create a parallel blend

1. When you choose **Done** from the BLEND OPTS menu, the system displays feature creation dialog box and the ATTRIBUTES menu. Choose either **Straight** or **Smooth**.
2. Create the first subsection using the Sketcher. You determine the direction of feature creation as you set up the sketching plane. Dimension and regenerate each subsection sketch to ensure the validity of the dimensioning scheme. A parallel blend requires more than one subsection, so after successfully regenerating this section, choose **Sec Tools** from the SKETCHER menu.
3. Choose **Toggle** from the SEC TOOLS menu. The first subsection turns gray and becomes inactive.
4. Choose **Sketch** and sketch the second subsection. Make sure its starting point corresponds to the starting point of the first by selecting the **Start Point** from the SEC TOOLS menu. Dimension and regenerate it. When you regenerate, the first subsection becomes active again.
5. If you are sketching more than two subsections, choose **Toggle** repeatedly until all the current geometry is gray, then sketch the subsection. Repeat this step until all subsections are sketched. Each subsection must be fully dimensioned to define its geometry and to locate it with respect to the other subsections. If you began your part with three default datum planes, every subsection can be dimensioned to them. Otherwise, each subsection should be dimensioned to another subsection or a local coordinate system (see the illustration Dimensioning Parallel Blend Sections).
6. To modify an existing subsection, toggle through until the subsection you want is active. While you can place or move the starting point of a subsection only when it is active, you can modify the dimensions of any subsection at any time.
7. When you have sketched all the subsections, choose **Done** from the SKETCHER menu. Enter the distances between each subsection in response to the prompts.
8. Select OK to create the feature. Pro/ENGINEER creates the blend feature in the direction of feature creation you specified when you set up the sketching plane.

Dimensioning Parallel Blend Sections



Projected Parallel Blend

Projected section blends allow you to create a sketch on a planar surface or datum plane and project the sections onto any two solid surfaces to create a blended feature. You cannot use external references when creating a projected section parallel blend.

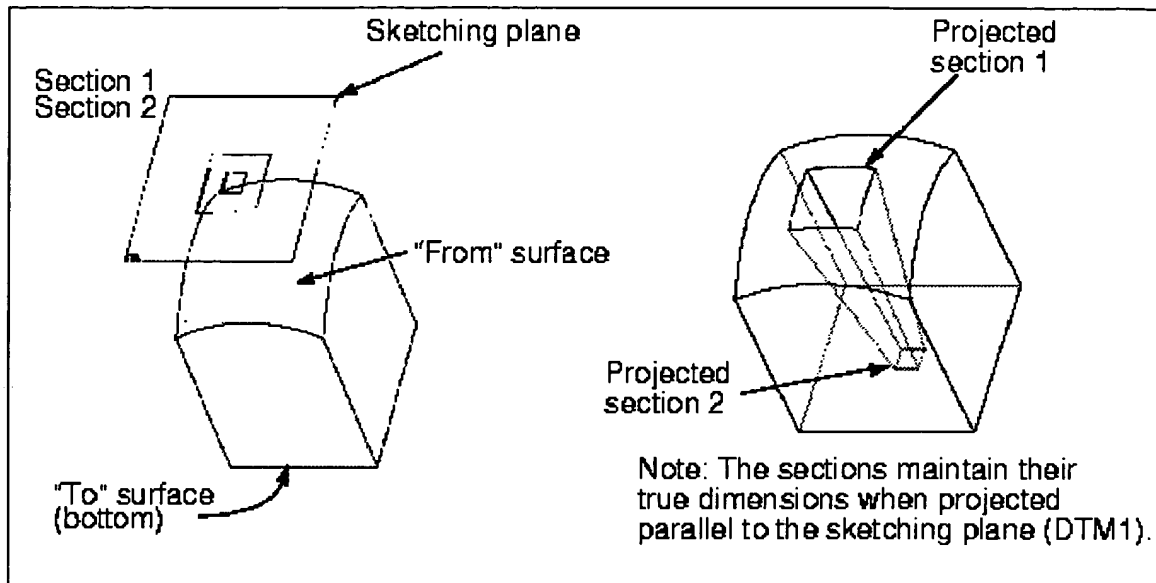
A projected parallel blend can have only two sections, each of which must lie within the boundaries of its selected surface, and cannot intersect other surfaces. When the sections are regenerated, the system projects them onto their selected surfaces, normal to the sketching plane (see the following illustration [Projected Section Blend](#)).

How to create a projected section blend

1. Choose **Project Sec** from the BLEND OPTS menu.
2. Select or create the sketching plane.
3. Select the "from" and "to" solid surfaces onto which the blend sections will be projected.
4. Sketch and dimension the two subsections, one for each surface, in the same order as you selected the **From To** surfaces (the first sketch will be projected onto the first surface selected).

[Projected Section Blend](#) illustrates a projected parallel blend.

Projected Section Blend



Non-Parallel Blends

Non-parallel blends (Rotational and General Options) have some particular advantages over parallel blends:

- Sections can be non-parallel, but do not have to be. Parallel blends can be created simply by entering a 0° angle between sections.
- Each blend section is created and saved in the database as a *separate* object.
- A section can be created by importing from an IGES file. See [Using IGES Files to Create Imported Sections for Non-Parallel Blends](#) for detailed information.

Sketched Versus Selected Sections

Non-parallel blend sections can be created by sketching them (using Sketch Sec), or by selecting three-dimensional entities (using the option Select Sec).

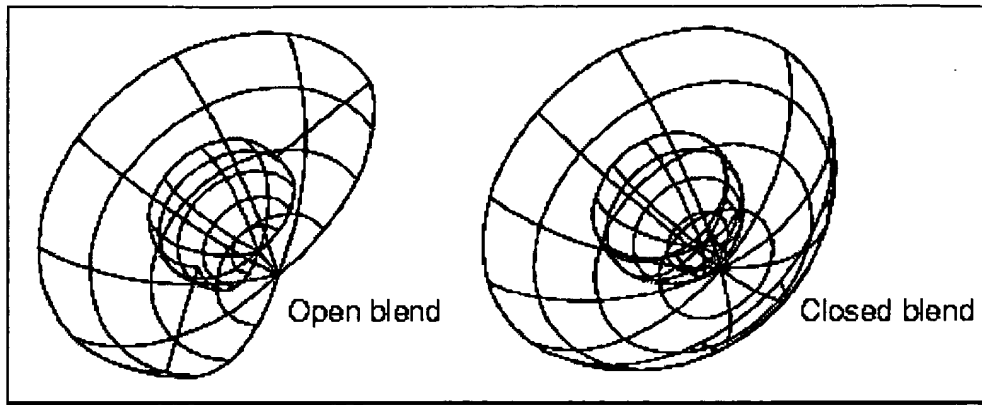
The restrictions for selecting section entities are as follows:

- All the entities must lie in the same plane.
- For rotational blends, the planes of all sections must intersect at a single axis. For rotational blends with only two sections, there is never ambiguity. However, if more than two sections are defined and they do not form a single axis, the feature fails and Pro/ENGINEER informs you that they do not have a common axis.

Open and Closed Blends

Non-parallel blends can be open or closed. If you specify Closed, Pro/ENGINEER uses the first section of the blend as the last section and creates a closed, solid shape.

Open and Closed Blends



Specifying Tangent Surfaces

You can create a smooth transition between the surfaces of a blend feature and surfaces of an adjacent feature on the same part. Open, smooth blends can have a tangent surface specified for each segment in the first and last sections. This means that the blend surface adjacent to this entity will be tangent to the selected surface.

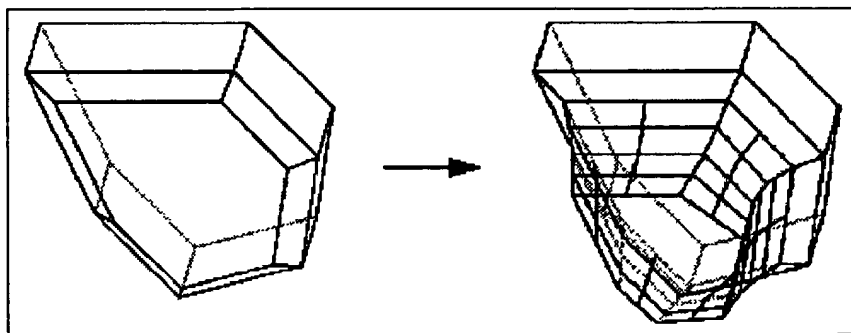
How to specify surface tangency conditions

1. When the system displays the SEL ELEMENT menu, choose **Tangency** and **Done**.
2. Once all the blend sections are created, the system asks if the blend should be tangent to any surfaces at the first end.
3. If you answer "yes", the system highlights each segment in the first section sequentially. Select a surface for each highlighted entity. If you do not want to specify tangency for the highlighted segment, choose **Done Sel** to move to the next segment.
4. Repeat the process for the other end of the blend.

For more information on specifying optional tangency, see [How to create a blended surface](#).

The following figure illustrates tangent surfaces.

Blend Tangent to Adjacent Surfaces



Using IGES Files to Create Imported Sections for Non-Parallel Blends

The following sections describe how to use IGES files to create non-parametric and parametric sections for non-parallel blends.

Non-Parametric Section

Importing an IGES feature during feature creation can produce a non-parametric feature that has no dimensions to modify interactively. You can use the imported feature to define a section.

How to use an imported feature to define a section

1. When creating a non-parallel blend, select or sketch a section (see [Sketched Versus Selected Sections](#)). Sketch a coordinate system and align the section to the part, then regenerate the section.
2. Choose **Interface** from the SEC TOOLS menu.
3. Choose **Import** from the INTERFACE menu and **IGES** from the INTF IMPORT menu.
4. Enter an IGES file name for a two-dimensional section. An Information Window displays a summary of the IGES import data.

Blends created in this way are subject to the following restrictions:

- Modifying the dimensioning scheme of the feature causes Pro/ENGINEER to prompt you for a new IGES file name. There must be a one-to-one correspondence between existing IGES entities and the replacement IGES entities (the first entity in the IGES file replaces the first entity in the section).
- The IGES entities are placed using their absolute coordinate values. There is no option to scale or dimension the resulting sketch.
- The IGES file section must be closed and all endpoints must be matched exactly with another endpoint. If an IGES file import fails, violating this restriction is the most likely cause.

Parametric Section

You can create a parametric section using an IGES file by importing the section into the Sketcher, dimensioning the entities, and regenerating the section. This type of section is more useful, because the resulting feature is fully parametric.

Functions Applicable to Both Types of Blends

This section describes how to use a blend vertex and capping blends.

Using a Blend Vertex

With the exception of capping a blend (see [Capping Blends](#)), each section of a blend must *always* contain the same number of entities. However, a blend surface can be made to disappear using a blend vertex on a sketched or selected section. A blend vertex acts as a terminator for the corresponding surface of the blend, but is counted in the total number of entities for a section. You can use a blend vertex in either a

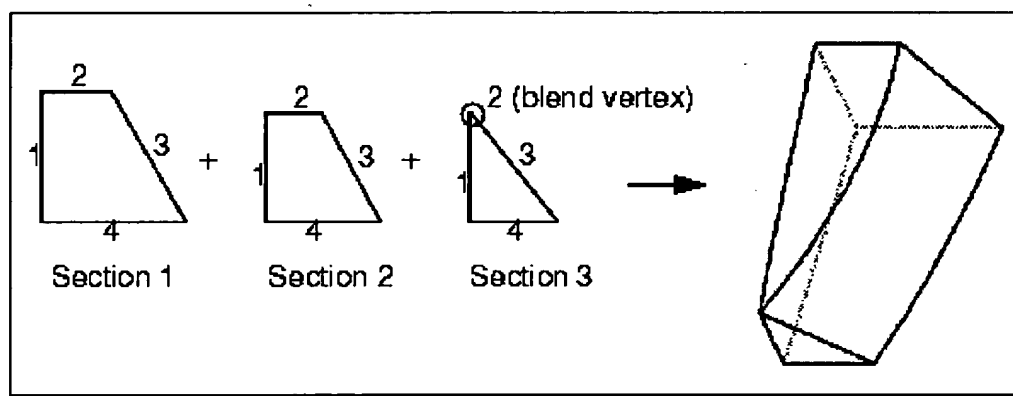
straight or smooth blend (including parallel smooth blends), but only in the first or last section.

How to add a blend vertex

1. Choose **Adv Geometry** from the GEOMETRY menu in Sketcher.
2. Choose **Blend Vertex** from the ADV GEOMETRY menu.
3. Select the vertex of an existing geometry entity. A circle will be placed there. More than one blend vertex can be created at the same point. Each additional vertex will create a concentric circle of increasing diameter, as shown in the following illustration, [Adding a Blend Vertex](#).

You can delete a blend vertex entity using Query Sel.

Adding a Blend Vertex

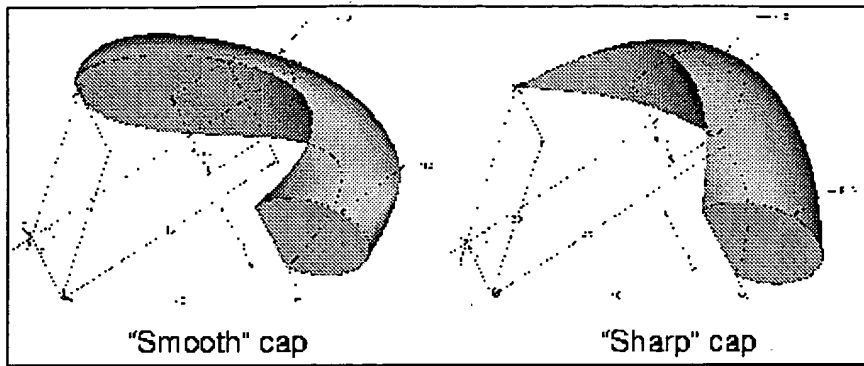


Capping Blends

The first and last sections of a blend can each be a zero-area section, that is, a point. This caps the end of the blend feature with either a sharp or smooth transition to a tip. The end subsection of a parallel blend can be defined using a single point. The end subsection of a parallel blend must always form a sharp cap.

Smooth and sharp caps create very different features, as shown in the following illustration, [Cap Type Affects the Feature Shape](#). The smooth cap is created by forcing all geometry to be tangent at the point section. The sharp cap allows the geometry to flow straight towards the point section. The best way to control the shape of the feature as it approaches the cap is to use as many sections as are necessary to achieve the desired result. However, when defined as a straight blend, or only as the second section of a non-parallel blend, the cap will be sharp regardless of what you select.

Cap Type Affects the Feature Shape



Note the following information about capped blends:

- The Z-axis is normal to the surface at the point entity. Entering rotation values for the X- and Y-axis affects the feature definition of a smooth cap.
- For a smooth cap, the point entity must be located within the boundaries of the previous section (picture where it would be if you used the same dimensions, but had sketched it on the previous section).

How to cap a non-parallel blend

1. For the last section of the non-parallel blend, create a coordinate system and a point entity. Dimension the point, if necessary.
2. Regenerate the section and choose **Done**.
3. The system displays the CAP TYPE menu, which has the following options:
 - **Smooth**-Create a cap that is smooth.
 - **Sharp**-Create a cap that is sharp.
4. Choose one of the options. Pro/ENGINEER creates the feature.

Rotational Blends

A rotational blend is created by sections that are rotated about the Y-axis. A rotational blend can contain up to 42 sections. You enter angular dimensions to control section orientation and can dimension sections from their Sketcher coordinate system to control radial placement.

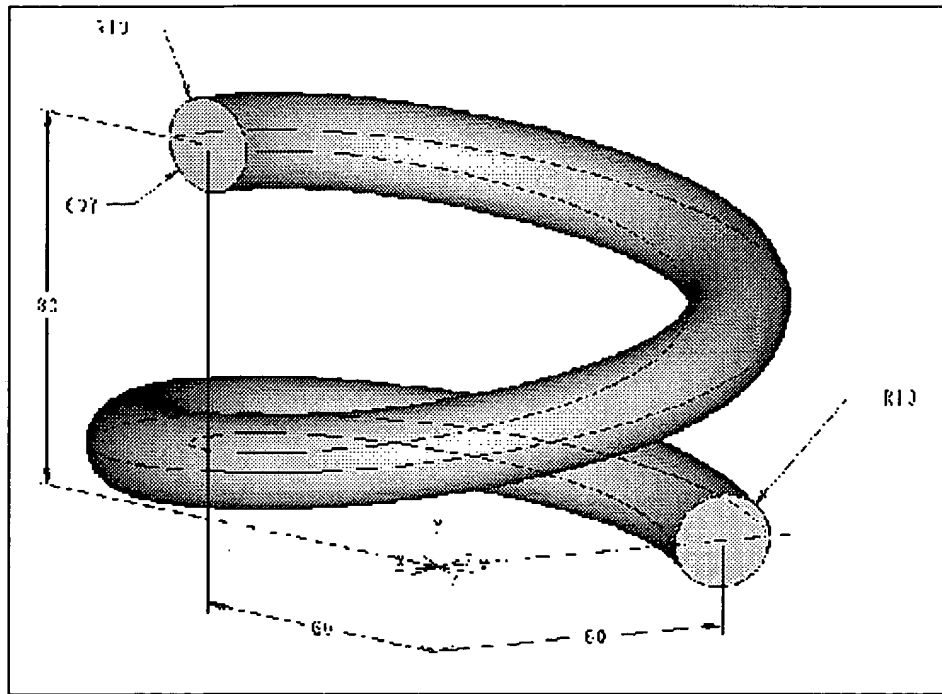
If you define a rotational blend as being closed, Pro/ENGINEER uses the first section as the last section and creates a closed solid feature. There is no need to sketch the last section.

How to create a rotational blend

1. When you choose **Rotational**, other options, and **Done** from the BLEND OPTS menu, the system displays feature creation dialog box with the required elements Attributes and Section. You can also choose the Tangency element if you want to specify optional tangency. When you have selected all the elements, select the Define button.

2. Choose from the mutually exclusive pairs of elements in the ATTRIBUTES menu, then choose **Done**. The possible choices are as follows:
 - **Straight**-Create a straight blend by connecting vertices of different subsections with straight lines. Edges of the sections are connected with ruled surfaces.
 - **Smooth**-Create a smooth blend by connecting vertices of different subsections with smooth curves. Edges of the sections are connected with spline surfaces.
 - **Open**-Create an open solid shape.
 - **Closed**-Create a closed solid shape. Pro/ENGINEER uses the first section of the blend as the last section.
3. Sketch or select the entities for the first section, including a Sketcher coordinate system (created using **Coord System** in the ADV UTILS menu). Use **Sketch Sec** to create the sections of the blend by sketching, or **Select Sec** to select three-dimensional entities. See Sketched Versus Selected Sections for more information.
4. For sketched sections, first enter the Y-axis rotation angle for the next section (120° maximum). After regenerating the section, the system displays a separate window for you to sketch the next section. After sketching and regenerating the section, choose **Done** from the Sketcher menu. The system prompts you whether to continue to the next section. If you reply "yes", repeat this step until you are done with all the sections.
5. If you are creating a smooth blend and selected **Opt Tangency** in the dialog box, create the blend with surfaces tangent to adjacent geometry. See Specifying Tangent Surfaces for more information.
6. When you have sketched or selected all sections, select **OK** in the dialog box to create the feature.

Sketched Rotational Blend with the First and Last Sections Displayed

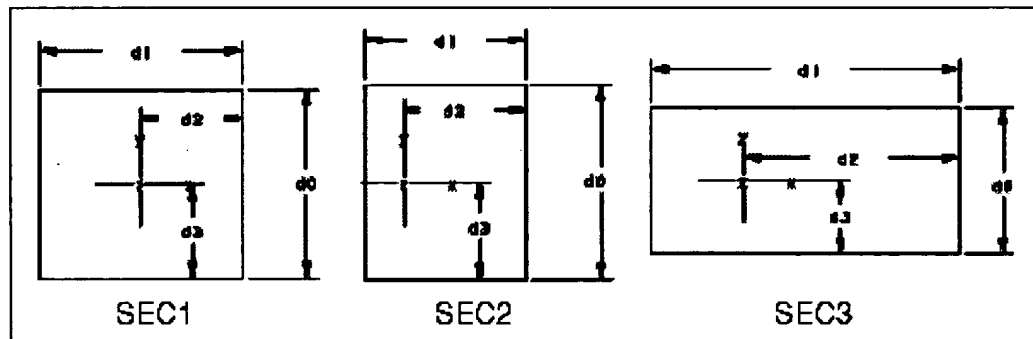


General Blend

How to create a general blend

1. When you choose **General** and **Done** from the BLEND OPTS menu, the system displays the feature creation dialog box and the ATTRIBUTES menu. Choose either **Straight** or **Smooth** from the ATTRIBUTES menu.
2. Sketch or select the entities for the first section, including a Sketcher coordinate system (created using **Coord System** in the ADV UTILS menu). Use **Sketch Sec** to create the sections of the blend by sketching, or **Select Sec** to select three-dimensional entities. See Sketched Versus Selected Sections for more information. For sketched sections, enter the X-, Y-, and Z-axis rotation angle (120° maximum) as prompted to determine the orientation of the next sketch, or reply "no" to the prompt (after the second section is defined) whether to continue to next section.

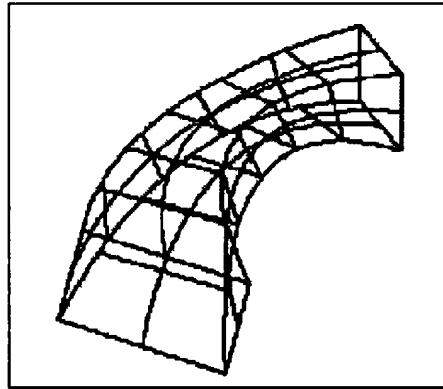
General Blend Sections



3. Repeat step 2 until you are done with all the sections.

4. After all the sections of the blend are finished, enter an offset depth value for all sections but the first. This dimension is the straight-line distance between coordinate system origins.
5. If you are creating a smooth blend and selected the **Opt Tangency** element in the dialog box, create the blend with surfaces tangent to adjacent geometry. See [Specifying Tangent Surfaces](#) for more information.
6. If you are creating a smooth blend, select tangency and section options. You can create the sections of the blend by sketching (using **Sketch Sec**), or by selecting three-dimensional entities (using **Select Sec**). See [Sketched Versus Selected Sections](#) for more information.
7. When you have sketched or selected all sections, select **OK** in the dialog box to create the feature.

General Blend



The Use Surfs Option

The Use Quilt option allows you to transform surface features into construction features (protrusion, cuts, and slots). See [Using Surfaces in Construction Features](#) for more information.

Advanced Form Features

The Advanced option in the Solid Opt menu accesses options for advanced features, some of which are combinations of blend and sweep functionality. The Adv Feat Opt menu contains the following options for creating swept, blended, and free-form features:

- **Var Sec Swp**-Display the Options menu for creating a variable section sweep, a feature using a single variable section.
- **Swept Blend**-Display the Options menu for creating a swept blend, a feature using multiple variable sections.
- **Helical Swp**-Create a sweep feature by sweeping a section along a helical trajectory.
- **Boundaries**-Create a blended feature using a number of curves as the boundary. See the section [Creating a Surface by Defining Its Boundaries](#) for detailed information on using this option.

- **Sect to Srfs**-Create a transitional surface between a set of tangent surfaces and a sketched contour.
- **Srfs to Srfs**-Create a smooth transition between two surfaces.
- **From File**-Create a blended feature by reading in data points from an ASCII file.
- **Free Form**-Dynamically "push" or "pull" on a surface, interactively changing its shape either to create a new surface feature, or modify a solid or quilt. For detailed information, see "[Freeform Manipulation](#)".

Variable Section Sweeps

If you have a Pro/FEATURE license, you can define a solid sweep feature using one or more longitudinal trajectories and a single variable section. The parameters of the section can vary as the section moves along the sweep trajectories. These sweeps are called variable section sweeps (formerly called "multi-trajectory sweeps"). With the optional Pro/SURFACE module, you can select trajectories instead of sketching them and you can create variable section sweeps of surfaces.

Every variable section sweep requires one longitudinal "spine" trajectory. You can define a sweep for which the X-axis of the section follows the X-vector trajectory, while remaining normal to the spine at all times as it sweeps along the spine (Nrm To Spine). To do so, you must also specify an X-vector trajectory to orient the section as it sweeps along the spine. The section plane is always normal to the spine trajectory at the point of their intersection. The X-axis of each section's coordinate system is defined by the direction from the point of intersection of the plane and the spine to the point of intersection of the plane and the X-vector trajectory for that section.

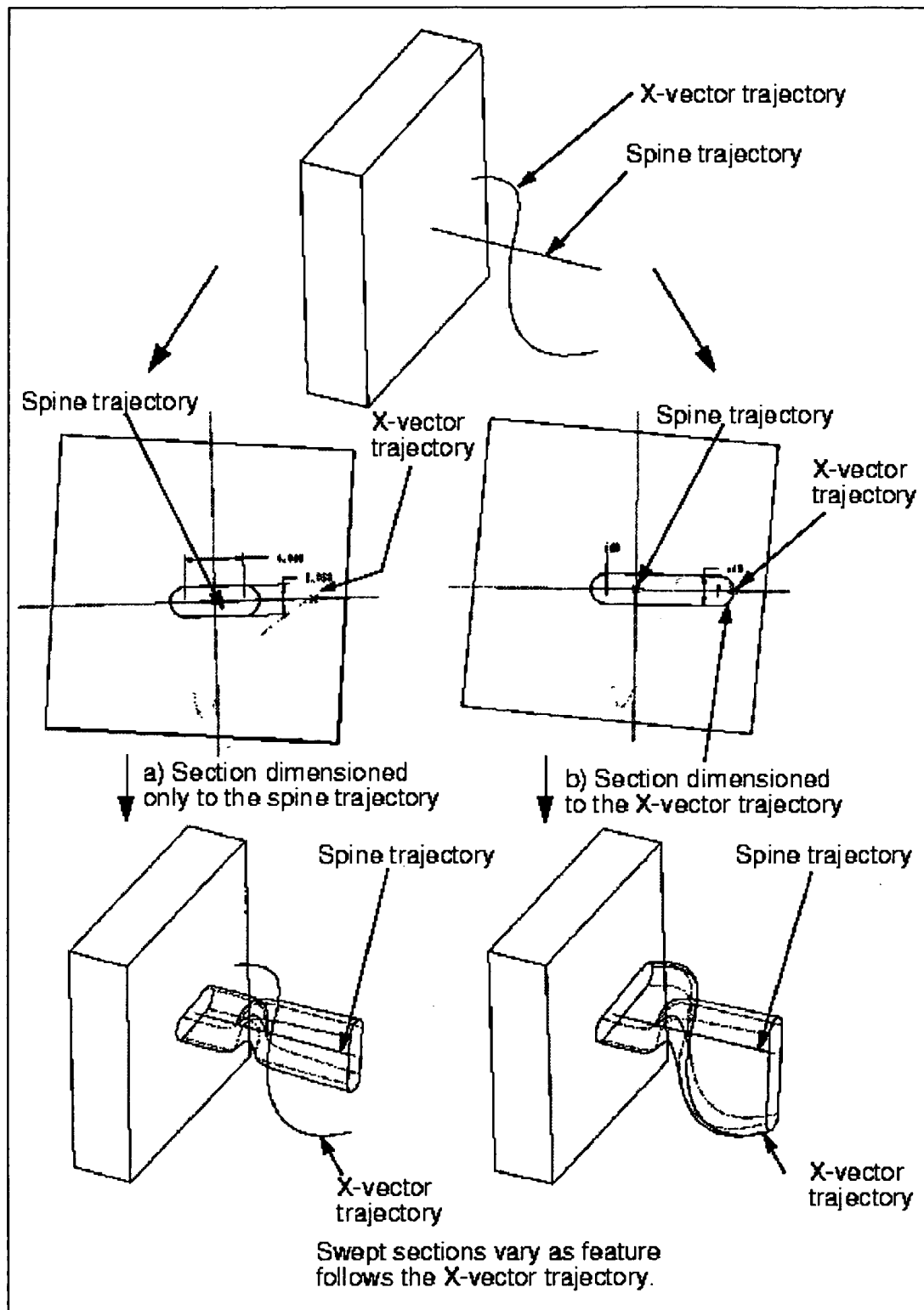
You can also define a variable section sweep for which the Y-axis of the section remains constant. The section will follow the spine such that it is normal to the selected pivot plane. The X-axis and Z-axis will still follow the spine and X-vector trajectories.

In summary, sweeps need the following trajectories:

- **Spine trajectory**-The trajectory along which the section is swept. If you choose Nrm To Spine, the origin of the section (crosshairs) is always located on the spine trajectory with the X-axis pointing towards the X-vector trajectory (as shown in the following figure).
- **X-vector trajectory**-Sweeps created using Nrm To Spine need this additional trajectory. It defines the orientation of the X-axis of the section coordinate system. *The X-vector and spine trajectories cannot intersect.*

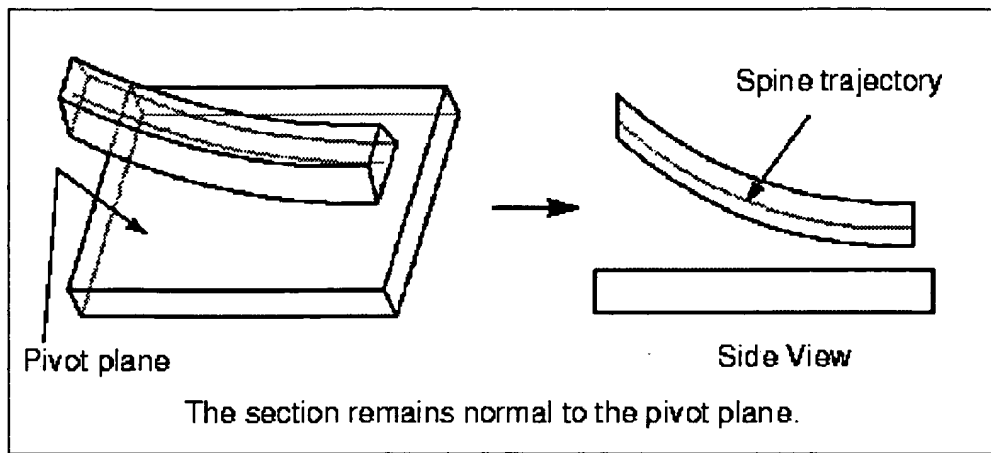
Once the direction is specified, the system displays the Var Sec Swp menu so you can define a trajectory. You can use a composite curve as a trajectory.

Creating a Variable Section Sweep with Nrm To Spine



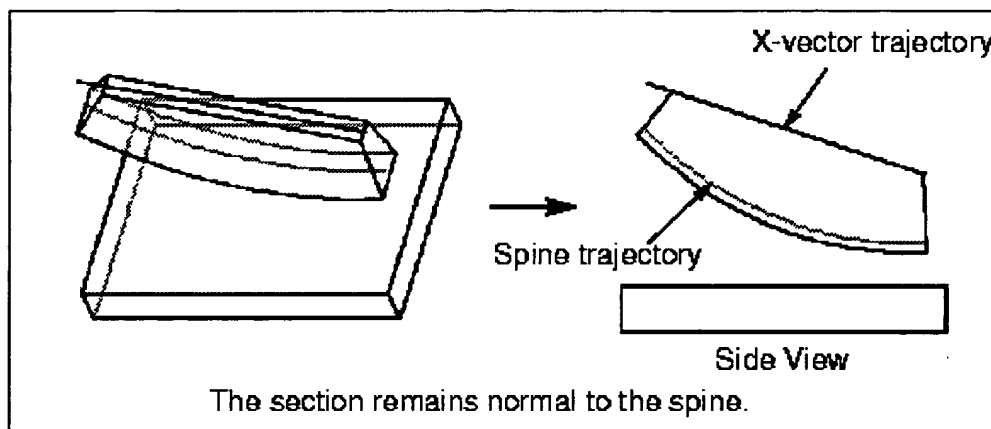
Pivot Dir Variable Section Sweep illustrates a variable section sweep that uses the Pivot Dir option.

Pivot Dir Variable Section Sweep



Nrm To Spline Variable Section Sweep illustrates a variable section sweep that uses the Nrm To Spline option.

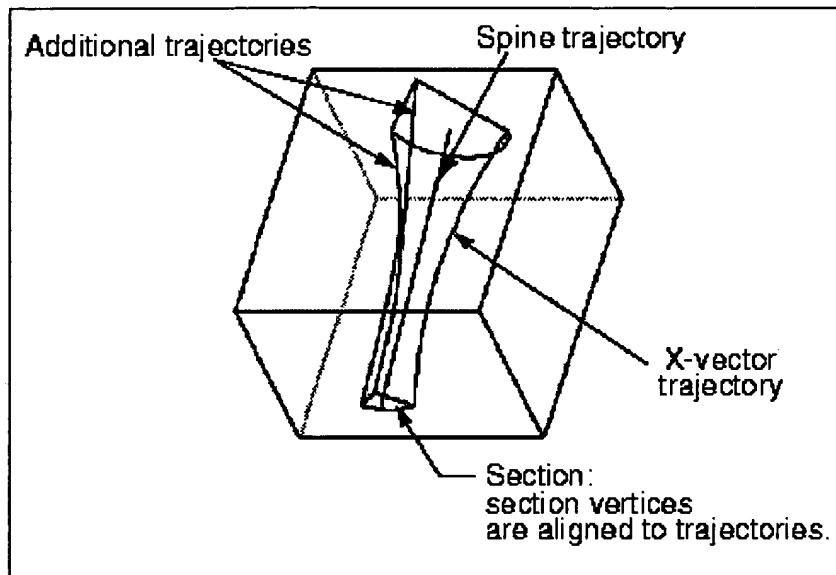
Nrm To Spline Variable Section Sweep



You can create any number of additional longitudinal curves that can be used to specify trajectories to which the vertices of the section can be aligned. As the section plane is swept along the spine, its intersections with the longitudinal curves represent the known points for section alignment and dimensioning. Illustrations Nrm To Spline Variable Section Sweep and Using Additional Trajectories show how you can affect the shape of the feature by dimensioning the section sketch to the X-vector trajectory. Section vertices are aligned or dimensioned to the X-vector trajectory and additional trajectories in Sketcher mode. Thus, each curve represents the trajectory of a section vertex as the section plane is swept along the spine.

Note that the radius of the arc in the section shown in the illustration Using Additional Trajectories is *not* changed throughout the feature unless you use relations to drive it. Relations are a powerful tool for creating the variable section sweep feature. To capture your design intent when sketching a variable section sweep section, you can use known dimensions (see Dimensioning Sections to a Part) and graph evaluation (for more information, see Relations in the *Fundamentals* manual). You can map a graph, or any function, along the variable section sweep spine using the trajectory parameter, "trajpar", in a relation. For additional information, see Using Relations in Sweeps.

Using Additional Trajectories



Restrictions

The restrictions on sweep trajectories (see [Rules for Defining a Trajectory](#)) also apply for variable section sweeps. Note the following additional restrictions:

- For Nrm To Spine sweeps, the spine trajectory can only consist of tangent entities. For Pivot Dir sweeps, projection of the entities must be tangent as viewed along the pivot direction (the entities themselves could be non-tangent in 3D).
- Either endpoint of the X-vector trajectory can meet the spine. However, the X-vector trajectory cannot cross the spine. If it does cross the spine, the section orientation at that point will be undefined and the system cannot construct the feature.
- All additional trajectories of the feature must intersect the sweep's sketching plane. The additional trajectories do not need to be as long as the spine trajectory; the sweep feature will be created as far as the endpoint of the shortest trajectory. Modifying the lengths of trajectories will modify the length of the sweep.
- Note that when you create a swept blend using a non-contiguous trajectory, you cannot add sections.

How to create a variable section sweep

1. Choose **Advanced** and **Done** from the SOLID OPTS or SRF OPTS menu. The system displays the ADV FEAT OPT menu.
2. Choose **Var Sec Swp**, then **Done** from the ADV FEAT OPT menu. Pro/ENGINEER displays the feature creation dialog box and the VAR SEC SWP menu.
3. Choose an SWEEP OPTS menu option, then choose **Done**. The possible options are as follows:
 - **Nrm To Spine**-The section plane remains always normal to the spine trajectory.

- **Pivot Dir**-The section plane pivots around the pivot direction such that the section's Y-axis remains constant while its X-axis and Z-axis follow the spine's projection. This causes the plane of the section to always remain normal to the spine trajectory, as viewed in projection along the pivot direction (see [Pivot Dir Variable Section Sweep](#)). The system displays the Gen Sel Dir menu to allow you to specify a pivot direction. The possible options are as follows:

- **Plane**-Select a plane or create a new datum plane to which the direction will be normal.
- **Crv/Edg/Axis**-Select as the direction an edge, curve, or axis. If you select a non-linear edge or curve, the system prompts you to select an existing datum point on the edge or curve to specify a tangent.
- **Csys**-Select an axis of the coordinate system as the direction. For additional information, see [Coordinate Systems](#).

4. Sketch or select the spine trajectory using the VAR SEC SWP menu options, then choose **Done**. The possible options are as follows:

- **Sketch Traj**-Sketch a new trajectory to use for the sweep.
- **Select Traj**-Define a chain from curves and edges (such as a datum curve) to use as the sweep trajectory.
- **Sel Tan Traj**-Define a chain from curves and edges to use as a trajectory and specify a tangency condition by selecting tangency reference surfaces (see [Specifying Tangency Conditions](#)).
- **Remove Traj**-Remove a trajectory that you previously sketched or selected.

5. If you chose **Pivot Dir** in Step 3, go to Step 6; otherwise, proceed as follows.

Sketch or select the X-vector trajectory. This defines the orientation of the section that will be swept by defining the horizontal vector of the section. Pro/ENGINEER sketches the sweep section on the normal plane located at a datum point, or the endpoint on the spine trajectory. The orientation of this sketch plane is such that the positive X direction (X-axis) passes from the endpoint of the sketched trajectory through a point at the intersection of the second trajectory with the normal plane. See [Chain Processing](#) for more information.

6. You can sketch or select as many additional longitudinal trajectories as you want, such that if the section is dimensioned to or aligned to these trajectories, the sweep feature also follows these trajectories as it travels along the spine. The trajectories can be accessed using the **Trajectories** option in the SEL ELEMENT menu. Choose **Done** to complete the trajectory definitions.
7. Sketch the sweep section. The section can be dimensioned to known points (to the points of intersection of the longitudinal curves with the sketching plane). Use relations to create a meaningful parametric section.

When you dimension the section to known points, consider the relative position of the X-vector trajectory throughout the spine. Dimensions that are valid at the start point of the sweep could

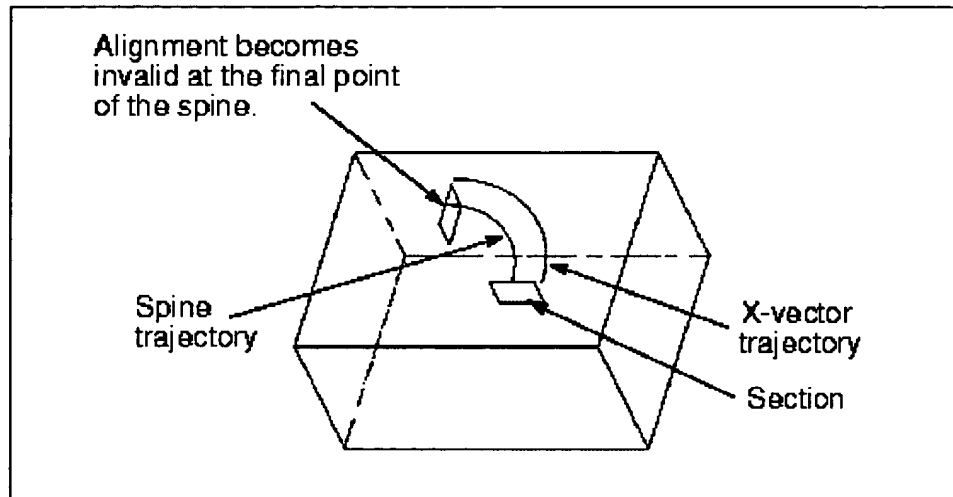
become meaningless as the section turns around the origin. The same consideration applies to aligning and dimensioning the section to part edges.

8. After the section sketch regenerates successfully, choose **Done**. To create the feature, select the OK button in the dialog box.

Aligning to Part Geometry

Consider the following recommendation: Do not align or dimension the section to part geometry unless the alignment or dimensions can be held throughout the sweep (see the following figure).

Aligning and Dimensioning a Variable Sweep Section



Specifying Tangency Conditions

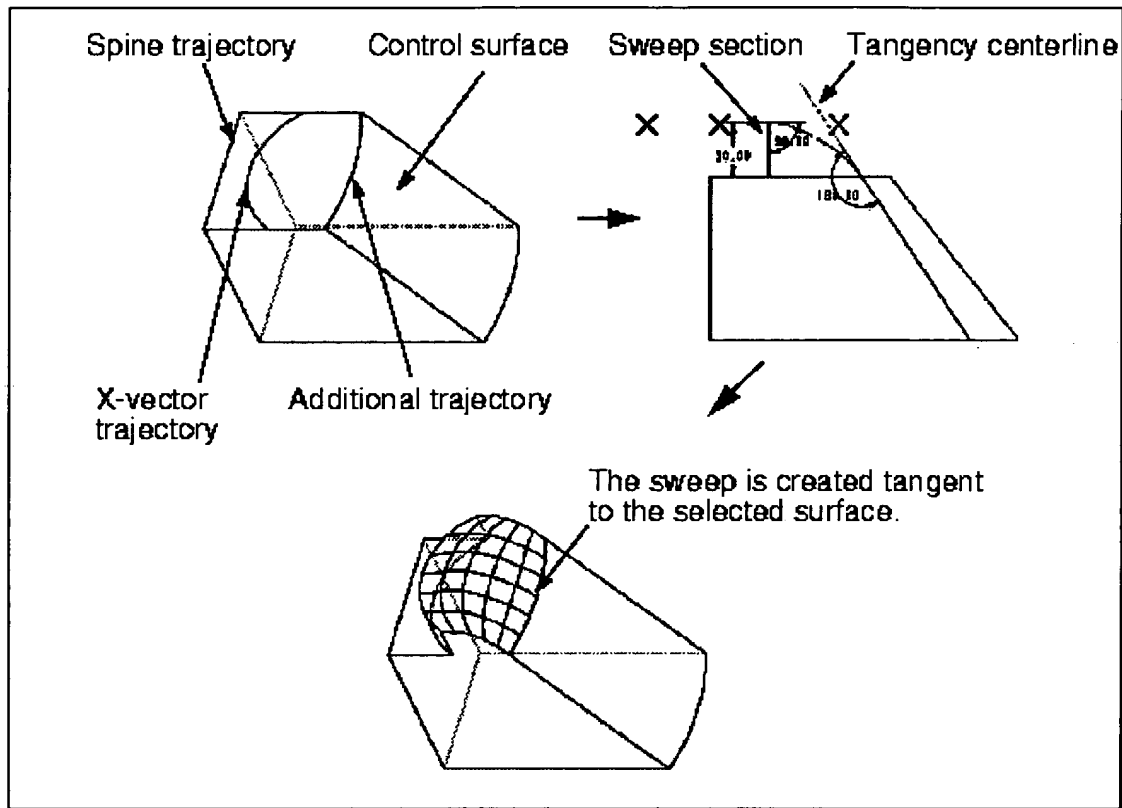
If you chose Sel Tan Traj to specify a tangent trajectory, you must specify a control surface for each segment of the trajectory. When the sweep section is created, the direction tangent to the control surface will be shown as a centerline and can be used for section dimensioning (see the following illustration, Sweep Tangent to a Surface).

How to specify tangency conditions

1. Once you selected the trajectory, the system highlights default tangent surfaces.
2. The system displays the DEFAULT TAN menu. Choose **Accept** to accept all the default surfaces, or choose **Reject** to select individual tangent surfaces.

When you start sketching the sweep section, all the specified tangencies are displayed as centerlines. You can use them for dimensioning in Sketcher mode; this way, a sweep surface can be forced to stay tangent to the adjacent part surface.

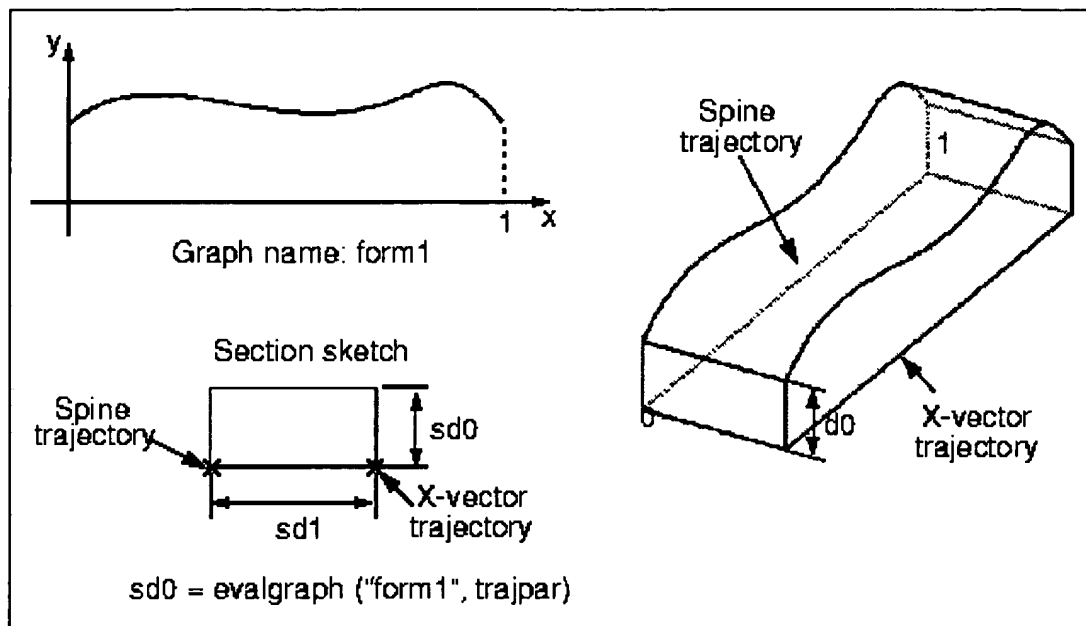
Sweep Tangent to a Surface



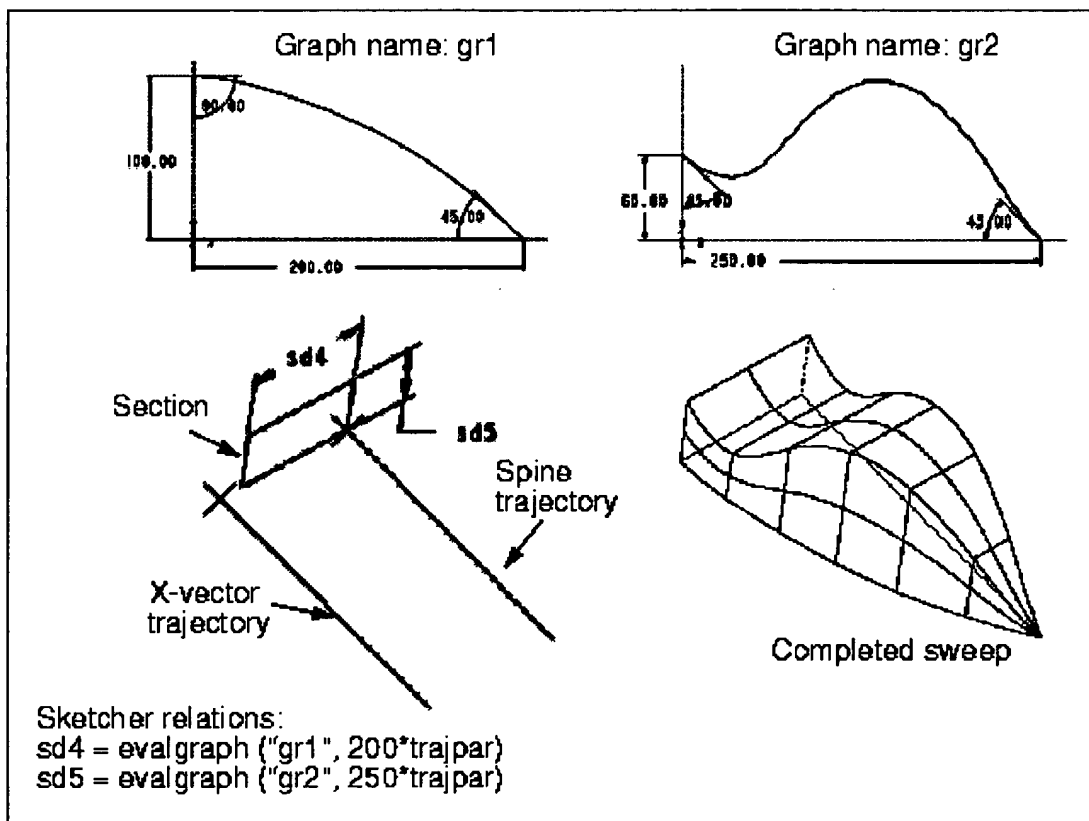
Using Relations in Sweeps

Using the trajectory parameter, *trajpar*, in a relation for variable section sweeps allows you to map a graph, or any function, along the sweep spine (see the following illustrations, [Mapping a Graph to a Variable Section Sweep](#) and [Variable Section Sweep Driven to a Point](#)). The value of *trajpar* changes from 0 to 1 as the section is swept along the spine. When a sweep is created along a composite curve, you can evaluate the *trajpar* of this curve at a specific point, the *trajpar_of_pnt* (see [Composite Datum Curves](#) and [Composite Curve Trajectory Function](#) in [Relations](#) of the *Fundamentals* manual), and use this value in relations. For an example, see [Parametric Graph Relations](#). If you set relations when sketching the section, connecting section dimensions with the trajectory parameter by some function, the section changes according to this function as it is swept along the spine.

Mapping a Graph to a Variable Section Sweep



Variable Section Sweep Driven to a Point

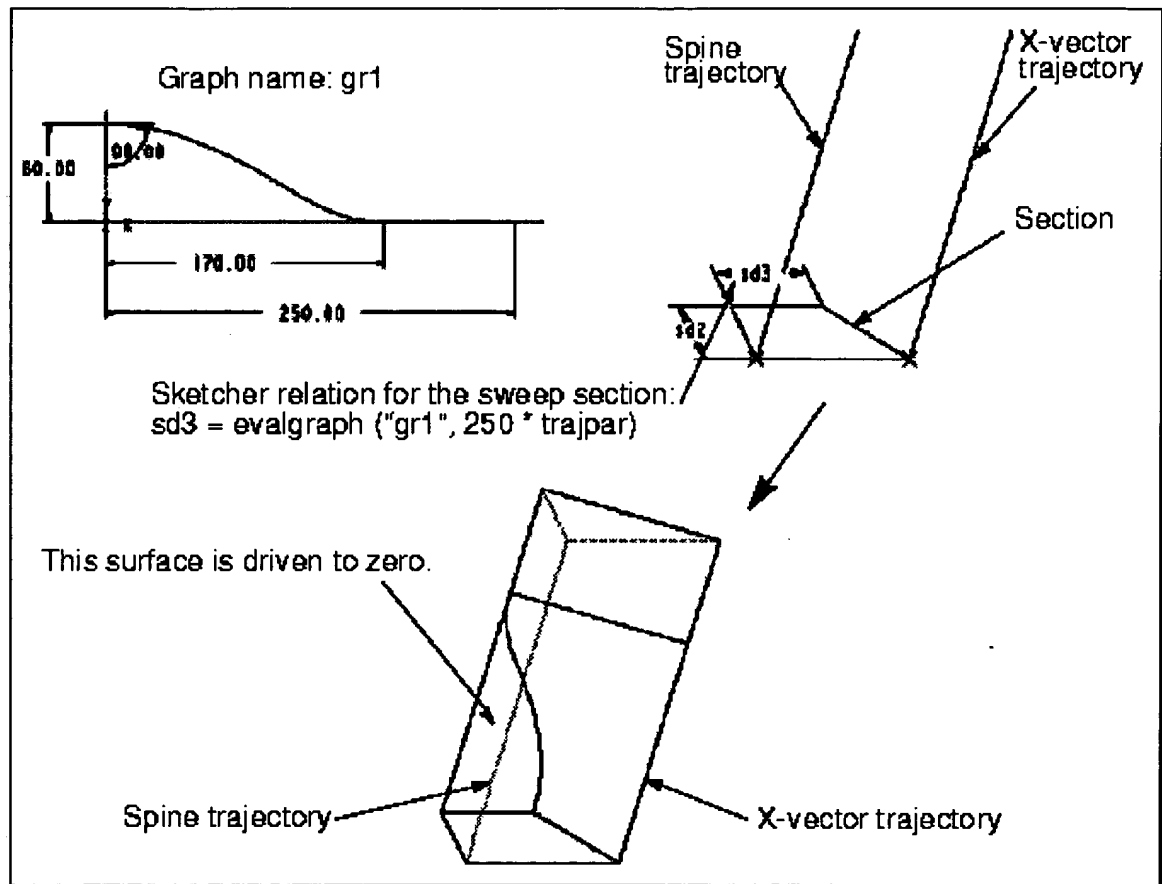


You can also create variable section sweep features with sections driven to zero area at the endpoint (see the preceding illustration, Variable Section Sweep Driven to a Point), or at some intermediate point of the trajectory of the feature (see the following illustration Spine Trajectory and Graph Relations).

Note:

The surface cannot be degenerate at the starting point of the trajectory. The dimension driven by a graph must evaluate to a non-zero value when you create the sweep section.

Spine Trajectory and Graph Relations



Parametric Graph Relations

If the driving graph is apt to change, you can include the dimensions of the graph instead of absolute values in the Sketcher relations of the sweep. The sweep then updates automatically with the changes to the driving graph.

How to create parametric graph relations

1. Before creating the sweep, choose **Feat Info** for the graph feature and determine the part dimension symbols corresponding to the appropriate section dimension of the graph.
2. When you dimension the sweep section, enter the relation for the corresponding graph dimension. In the previous example, the relation would be as follows:

`sd3 = evalgraph ("gr1", d0 * trajpar)`

Modifications to the graph function may change its shape so the sweep feature geometry cannot be created. If the sweep feature fails because of modifications to the driving graph, you have two options:

- Choose Sys Recover to restore all the dimensions of the graph.

- Choose **Trim Part and Redefine** to redefine the sweep trajectories and section. For more information, see [Regeneration Information](#).

Swept Blends

A swept blend is created using a single trajectory (a spine) and multiple sections. You create the spine of the swept blend by sketching or selecting a datum curve or an edge. You sketch the sections at specified segment vertices or datum points on the spine. To orient a section, you can specify the rotation angle about the Z-axis, and/or use the **Pick XVector** or **Norm to Surf** options.

Note the following restrictions:

- A section cannot be located at a sharp corner in the spine.
- For a closed trajectory profile, sections must be sketched at the start point and at least one other location. Pro/ENGINEER uses the first section at the endpoint.
- For an open trajectory profile, you must create sections at the start and end points. There is no option to skip placement of a section at those points.
- Sections cannot be dimensioned to the model, because modifying the trajectory would invalidate those dimensions.
- A composite datum curve cannot be selected for defining sections of a swept blend (**Select Sec**). Instead, you must select one of the underlying datum curves or edges from which a composite curve is determined.
- If you choose **Pivot Dir** and **Select Sec**, all selected sections must lie in planes that are parallel to the pivot direction.
- You cannot use a nonplanar datum curve from an equation as a swept blend trajectory.

If you have a Pro/SURFACE license, you can control swept blend geometry using an area graph. An area graph represents the exact area of the cross-section of the swept blend at selected locations on the spine. You can add or remove points on the spine at which to specify the swept blend sectional area. You can also change the graph value at user-defined points (see the illustration [Sample Area Graph and Information Window](#)). You choose how the section will be defined using the **Nrm To Spine** and **Pivot Dir** options.

Creating a Swept Blend

To create a swept blend, you can define the trajectory by sketching a trajectory, or by selecting existing curves and edges and extending or trimming the first and last entity in the trajectory.

How to create a swept blend

1. Choose **Advanced** from the **SOLID OPTS** menu, and **Swept Blend** and **Done** from the **ADV FEAT OPT** menu.

2. Choose the desired options from the BLEND OPTS menu mutually exclusive pairs, then choose **Done** from the BLEND OPTS menu. The possible options are as follows:
 - **Select Sec**-Select existing curves or edges to define each section using the CRV SKETCHER menu.
 - **Sketch Sec**-Sketch new section entities to define each section.
 - **Nrm To Spine**-The section plane remains normal to the spine trajectory.
 - **Pivot Dir**-The section plane will remain normal to the spine trajectory when viewed from a pivot direction. As the section plane sweeps along the trajectory, it always remains parallel to the pivot direction and the section pivots around the pivot direction.
3. A Swept Blend dialog box appears with the following elements:
 - **Spine**-Select the spine.
 - **Sections**-Define the sections.
 - **Blend Control**-(Optional) Define how to control the blend geometry along the spine.
 - **Tangency**-(Optional) Specify tangency conditions for the feature.
4. Define the type of spine by choosing an option from the SWEEP TRAJ menu:
 - **Sketch Traj**-Sketch a spine. The spine can have sharp corners (a discontinuous tangent to the curve), except at the endpoint of a closed curve. At non-tangent vertices, Pro/ENGINEER mitres the geometry as in constant section sweeps (see Swept Feature Corners).
 - **Select Traj**-Define the spine trajectory using existing curves and edges. The system displays the CHAIN menu (for details, see Chain Processing). Choose **Select**, define the chain, then choose **Done**.
5. The system highlights endpoints. Use options in the CONFIRM menu to select an endpoint for which you want to specify the section.
 - **Accept**-Sketch or select a section at this highlighted location.
 - **Next**-Go to the next point.
 - **Previous**-Return to the previous point.
6. If you selected the **Nrm To Spine** option, the system brings up the SEC ORIENT menu. Select one of these options, followed by **Done**:
 - **Pick XVector**-Select an axis, straight edge/curve, or plane normal to determine the section's positive axis. Use options in the GEN SEL DIR menu to select a horizontal reference. The system displays a red arrow, indicating the positive direction for the X-vector. Choose **Flip** or

Okay to determine the direction for the operation.

Note:

The Pick XVector option is available only for the trajectories defined with the Select Traj option.

- **Automatic**-The system automatically determines the section's orientation.

If you select this option for the first section, then the X-axis is determined by the curvature vector at the beginning of the spine. To specify the positive direction of the X-axis, choose **Flip** or **Okay**.

When you select **Automatic** for a section other than the first, the system determines the X-vector automatically based on the previous section orientation and the behavior of the spine.

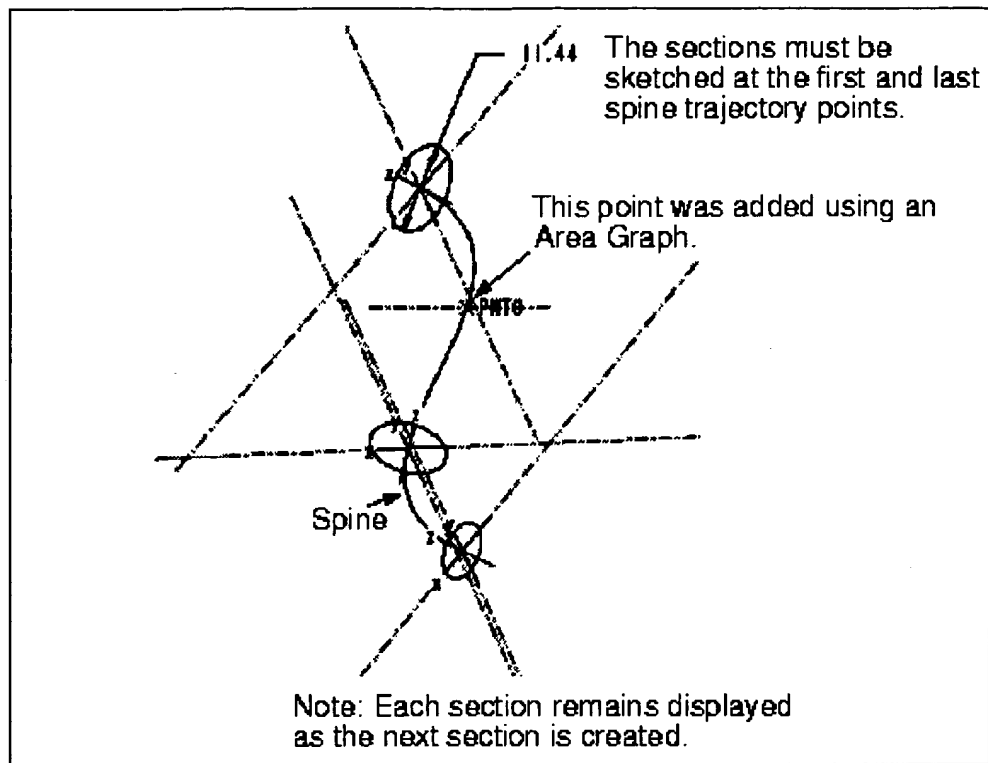
- **Norm to Surf**-Use the section normal to determine the section upward direction. If you select this option for the first section, then all sections use the same reference surface as the horizontal reference.

If the spine has only one adjacent surface, then the system automatically selects this surface, highlighted in blue, as the reference for the section orientation. A red arrow appears, indicating the upward direction. Choose **Flip** or **Okay** to specify the upward direction.

If the spine has two adjacent surfaces, the system prompts you to select a surface for the section orientation. The default surface is highlighted in blue. You can accept the default surface or select the other one. A red arrow appears, indicating the upward direction. Choose **Flip** or **Okay** to specify the upward direction.

7. For each section, specify the rotation angle about the Z-axis (with a value between -120 and +120 degrees).

Section Definition



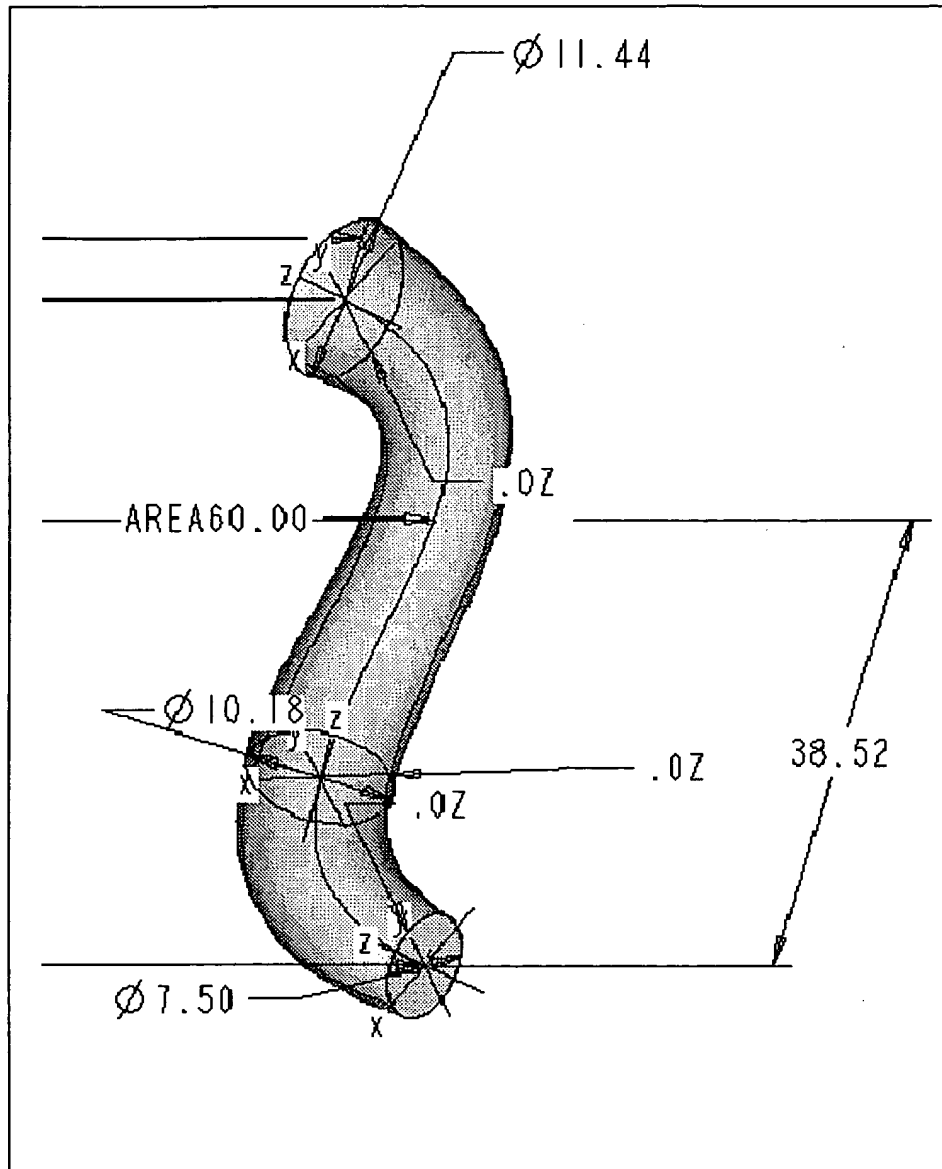
8. Select or sketch the entities for each section, depending on whether you chose **Select Sec** or **Sketch Sec**, respectively. Choose **Done** from the SKETCHER menu.

Note:

Dimension or align the section to the coordinate system, not part geometry.

9. When all cross-sections are sketched or selected, unless you want to define optional elements, select **OK** in the dialog box to generate the swept blend feature. If you want to define optional elements, continue as described in the following sections.

Completed Swept Blend



Modifying Swept Blend Geometry Using an Area Graph

The Define Graph menu allows you to add or remove control points on the spine at which you can specify or change area values.

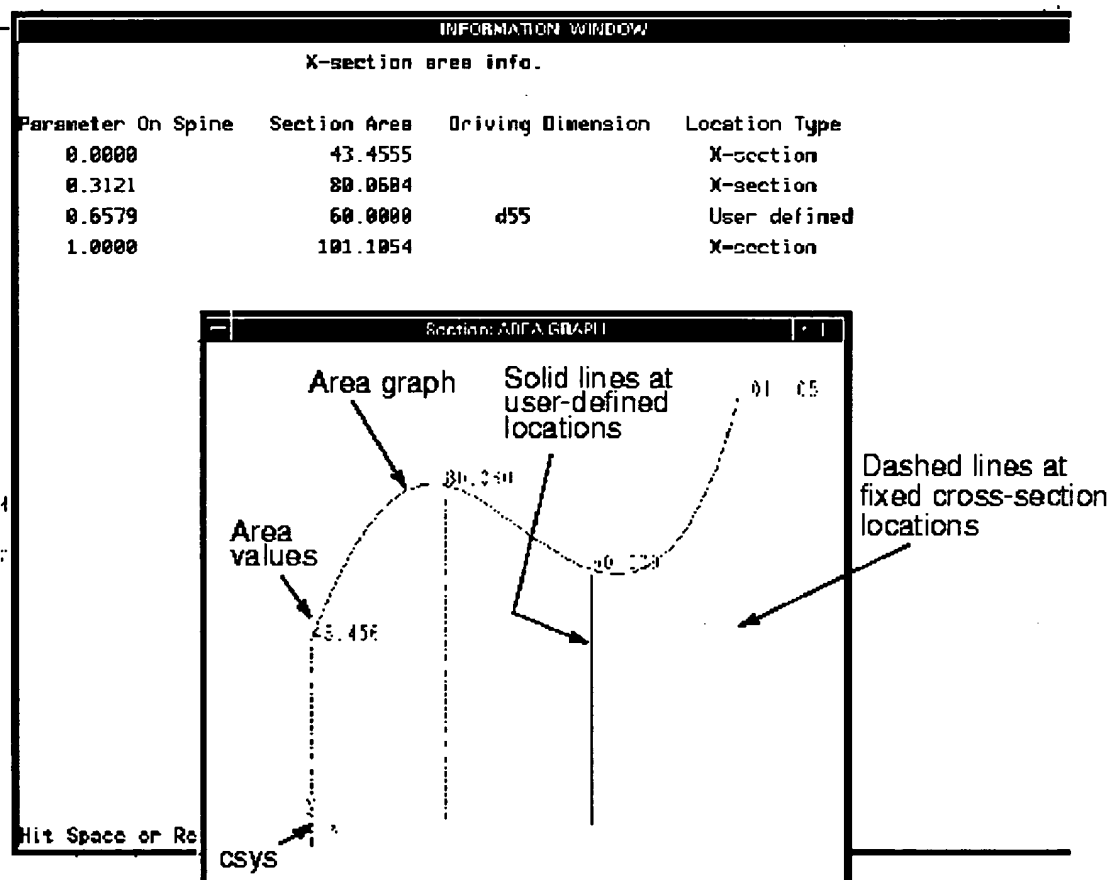
The Graph menu options are as follows:

- **Define**-Define an area graph using the Define Graph submenu. The Define Graph submenu options are as follows:
 - **Add Point**-Define a control point using the Get Dtm Pnt submenu to select or create a datum point on the spine, then enter the area values.
 - **Remove Point**-Select a control point to remove.
 - **Change Value**-Select a control point and enter a new area value. If a value is zero on the area

- **Change value**-Select a control point and enter a new area value. If a value is zero on the area graph at a parameter, the swept blend self-intersects. To correct this, add control points to change the area graph value to a positive value.
- **Info**-Display an Information Window (see the following figure), which contains the following information:
 - The normalized length of a parameter (point or cross-section) measured from the starting point of the current segment of the spine, in the form *i.rrrr*. A spine consists of one or more segments. The integer, *i*, identifies which of the segments on the spine the parameter is located. The value of *i* ranges from 0 to *n*, where 0 corresponds to the first segment and *n* to the last segment. The decimal .*rrrr* is the ratio of the length from the starting point on the segment to the parameter location.
 - The section area values at each parameter.
 - The driving dimension, if any, for the value of a user-defined area.
 - The location type specifies whether the area is at a section or at a user-defined point.

To redefine the area values and the control points using the AreaGraph interface, use the options Redefine, References.

Sample Area Graph and Information Window



Controlling the Perimeter of the Swept Blend

If you have a license for the Pro/FEATURE for BODY ENGINEERING module, you can use the Set Perimeter functionality to control the perimeter of a swept blend between its sections. When this option is available, the Blend Control element replaces the Area Graph element in the dialog box.

The Blend Control element lets you select a method for controlling the shape of the swept blend between its sections. When you choose Blend Control and Define from the dialog box, the Blend Cont menu appears with the following options:

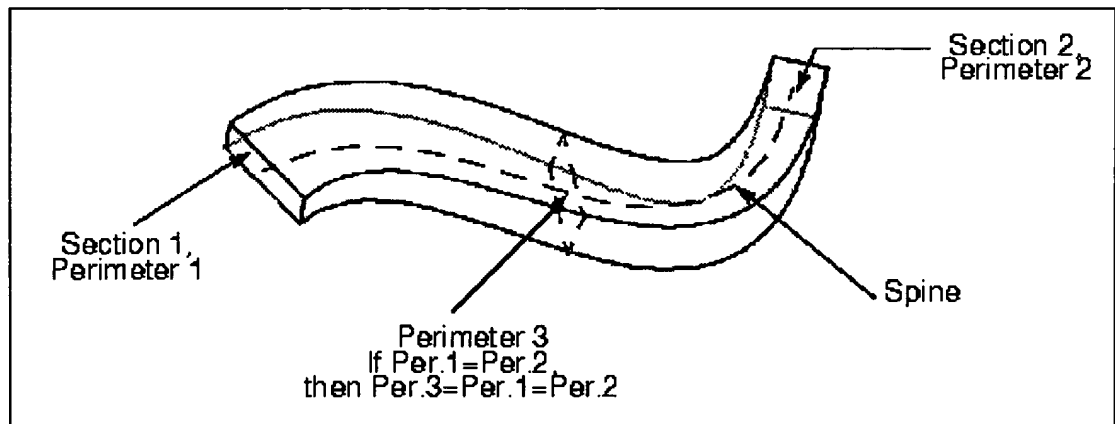
- **Set Perimeter**-Control the shape of the feature by controlling its perimeter between the sections. If two consecutive sections have equal perimeters, the system attempts to maintain the same cross-section perimeter between these sections (see the following illustration, [Using the Set Perimeter Option](#)). For sections that have different perimeters, the system uses smooth interpolation along each curve of the trajectory to define the perimeter of the feature between its sections.

Note:

You cannot specify both perimeter control and tangency conditions for the swept blend-only one of these conditions is allowed.

- **Area Graph**-Control the shape of the feature through control points and area values (see [Modifying Swept Blend Geometry Using an Area Graph](#)).
- **None**-Do not set any blend control for the feature.
- **Center Crv**-Show a curve connecting the centroids of the feature's cross-sections. This option is available only with the Set Perimeter option.

Using the Set Perimeter Option



Helical Sweep

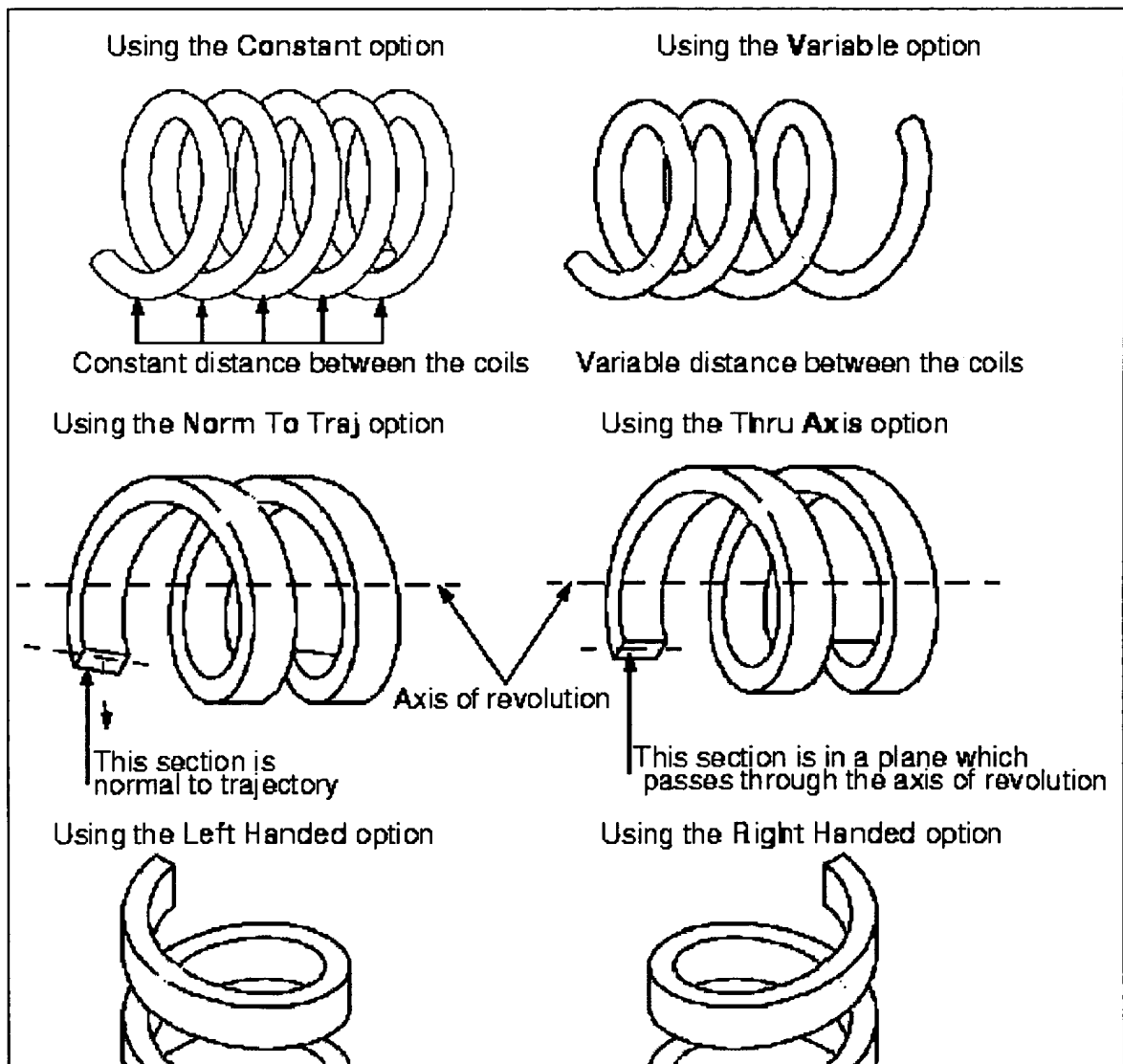
You create a helical sweep by sweeping a section along a helical trajectory. The trajectory is defined by both the "profile" of the surface of revolution (which defines the distance from the section origin of the helical feature to its axis of revolution) and the "pitch" (the distance between coils). The trajectory and the surface of revolution are construction tools that do not appear in the resulting geometry. See the following

illustration Types of Helical Sweep Features for an example of the different types of helical sweep features.

The Helical Swp option in the Adv Feat Opt menu is available for both solid and surface features. Use the following Attributes menu options in mutually exclusive pairs to define the helical sweep feature:

- **Constant**-The pitch is constant.
- **Variable**-The pitch is variable and defined by a graph.
- **Thru Axis**-The cross-section lies in a plane that passes through the axis of revolution.
- **Norm To Traj**-The cross-section is oriented normal to the trajectory (or surface of revolution).
- **Right Handed**-The trajectory is defined using the right- hand rule.
- **Left Handed**-The trajectory is defined using the left-hand rule.

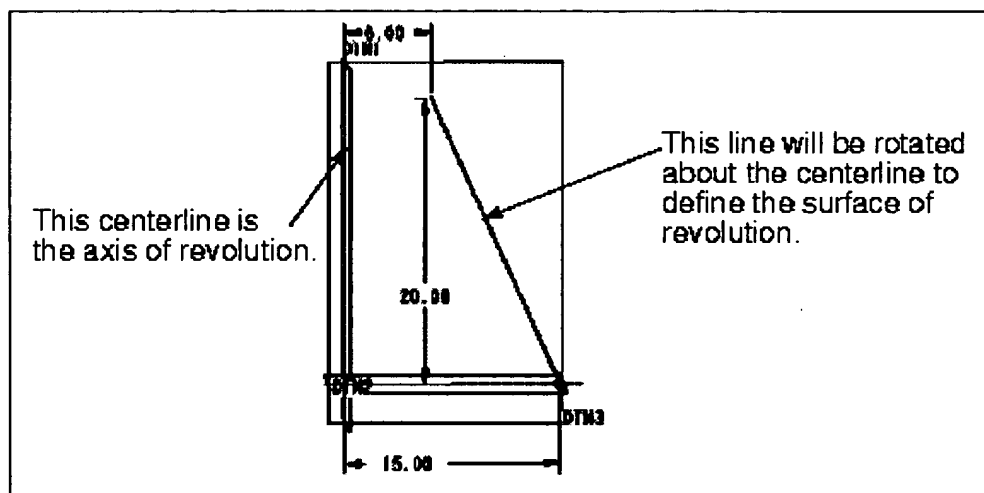
Types of Helical Sweep Features



How to create a helical sweep with constant pitch value

1. Choose **Advanced** and **Done** from the SOLID OPTS menu, then **Helical Swp** and **Done**. The system displays the feature creation dialog box.
2. Define the feature by selecting from the ATTRIBUTES menu, then choose **Done**.
3. Pro/ENGINEER places you in Sketcher mode. Sketch the profile of the surface of revolution. Specify the sketching plane and its orientation, and the axis of revolution.
4. Sketch, dimension, and regenerate the profile (see the following illustration, Profile for a Helical Sweep). Follow these rules:
 - The sketched entities must form an open loop.
 - You must sketch a centerline to define the axis of revolution.
 - If you chose **Norm To Traj**, the profile entities must be tangent to each other (*CI* continuous).
 - The profile entities should not have a tangent that is normal to the centerline at any point.
 - The profile starting point defines the sweep trajectory starting point. You can modify the starting point using the options **Sec Tools** and **Start Point**.

Profile for a Helical Sweep

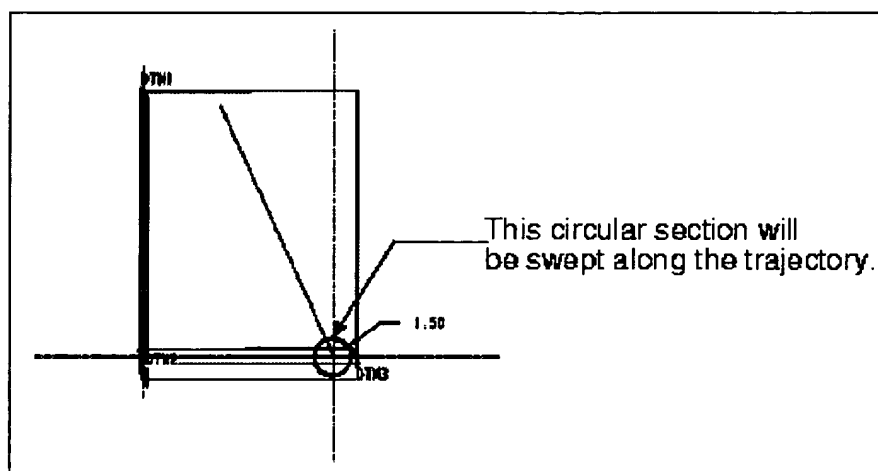


5. When you have finished sketching the section, choose **Done** from the SKETCHER menu.
6. Enter the pitch value (the distance between the coils).
7. For a surface feature, specify if the feature will have closed or open ends by selecting **Open Ends** or

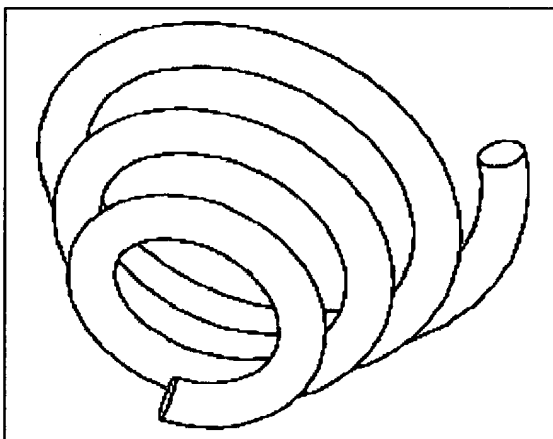
Capped Ends from the SURF END menu (see Open or Closed Ends), then **Done**.

8. Pro/ENGINEER places you in Sketcher mode to sketch the cross-section that will be swept along the trajectory. Sketch the cross-section based about the visible cross hairs. Dimension and regenerate the cross-section (see the following illustration, Cross-Section of the Helical Sweep).
9. When the cross-section is finished, choose **Done** from the SKETCHER menu. See Helical Sweep Feature with Constant Pitch for an illustration of the resulting feature.

Cross-Section of the Helical Sweep



Helical Sweep Feature with Constant Pitch



Variable Pitch Helical Sweeps

You can also create a helical swept feature with a variable pitch. In this case, the distance between the coils is controlled by a pitch graph. The initial graph (see the following illustration, Initial Pitch Graph) is created when you specify the pitch value at the start and end points. You can then add more control points to define a complex curve that governs the distance between the coils along the axis of revolution.

Special considerations for using the Variable option are as follows:

- In a pitch graph, control points with different pitch values are connected by a monotonic curve.

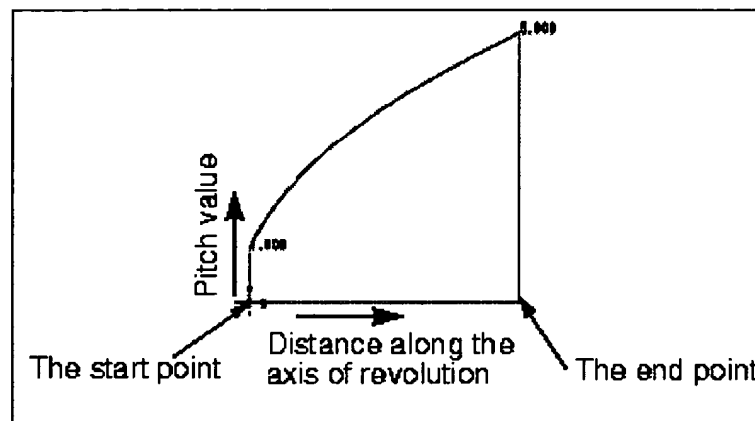
Control points with equal pitch values are connected by a line.

- In the resulting geometry, the average distance between coils along each portion of the axis (the segment between two control points in the pitch graph) is the average of the pitch values given at two consecutive control points.

How to create a helical sweep with a variable pitch value

1. Complete Steps 1 through 4 from the procedure How to create a helical sweep with constant pitch value.
2. While in the profile section, sketch points to be used as the control points in the pitch graph. These control points define how the pitch value changes along the axis of revolution. To sketch points, choose **Sketch, Point**, then select points on the profile geometry and dimension them. It is easier to dimension the control points if you put them on the centerline that defines the axis of revolution.
3. After you regenerate your profile sketch successfully, choose **Done** from the SKETCHER menu.
4. Enter pitch values at the trajectory start and end.
5. While the profile section is displayed in the original window, the system displays a subwindow with the initial pitch graph in it (see the following figure).

Initial Pitch Graph



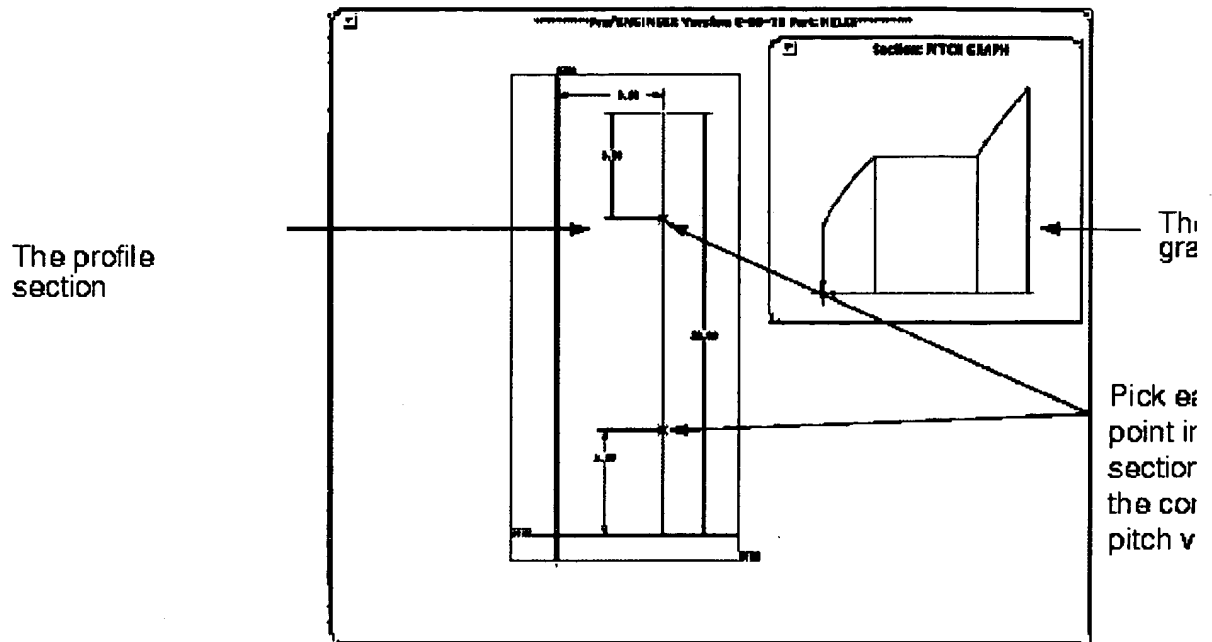
6. Finalize the graph by transferring the pitch control points from the profile sketch onto the graph (see the following illustration, Finalizing the Pitch Graph). Choose **Define** from the GRAPH menu.

Using options in the DEFINE GRAPH menu, do one of the following:

- **Add Point**-Add a reference point to the graph by selecting a point in the *profile* section, or the start or end point. Enter the desired pitch value at this point. The system locates the selected control point along the X-axis of the graph and draws a line with the length equal to the specified pitch value.
- **Remove Point**-Remove a pitch control point by picking it in the *profile* section.
- **Change Point**-Change the value of the pitch at any selected control point, including the start

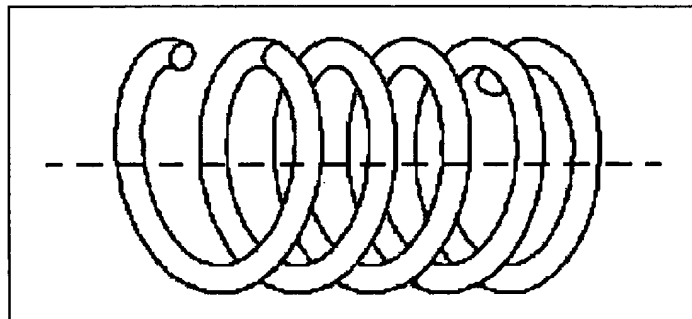
or end point. Select a point in the *profile* section to change its value and enter the new value.

Finalizing the Pitch Graph



7. Once the graph is defined, choose **Done/Return** from the DEFINE GRAPH menu. To check the graph data, choose **Info** in the GRAPH menu. The system displays the Information Window with the pitch data table.
8. Choose **Done** from the GRAPH menu.
9. Pro/ENGINEER places you in Sketcher mode to sketch the cross-section that will be swept along the trajectory. Sketch, dimension, and regenerate the cross-section.
10. When you have finished, choose **Done**. The resulting feature is shown in the following figure, Helical Sweep Feature with Variable Pitch.

Helical Sweep Feature with Variable Pitch

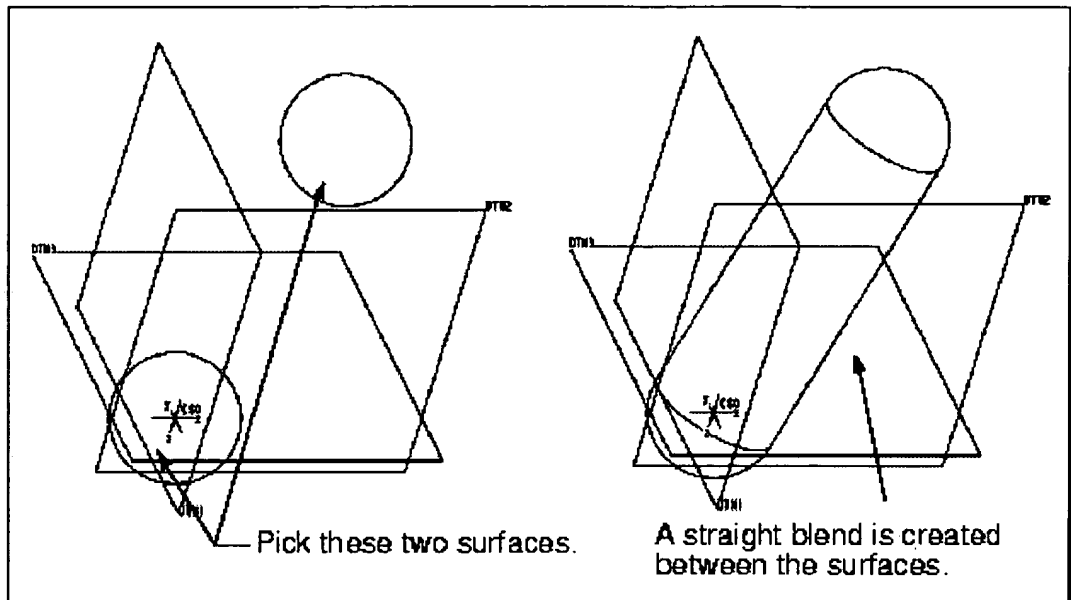


Section-to-Surface Blends

If you have the optional Pro/SURFACE module, the Adv Feat Opt option Sect to Srf's allows you to create a transitional surface between a set of tangent surfaces and a sketched contour. This option can be used for

•

Surfaces-to-Surfaces Blend



Importing Blends

Blends can be created by reading in data points from an ASCII file. The data file defines the type of blend, as well as the Cartesian coordinates of all the blend section points. All blend section points are located relative to a single coordinate system.

If you are importing data points from a measuring device, you should import them as curves first to insure smoothness. There are several techniques in Pro/ENGINEER for smoothing imported curves. You can then create a blended surface from the smoothed curves.

Blend File Format

The imported blend data file, with the file extension *.ibl*, has the following format:

```
/* beginning of file */  
section_type    /* The section type (open or closed). */  
blend_type      /* The blend method (arclength or pointwise).
```

Both blend types require the same number of curve segments for each section. An arclength blend uses a general blending routine to connect the sections. The number of points in corresponding curves can be different for each section. A pointwise blend connects from point to point (point 1 in one curve connected to point 1 in the other curve). The corresponding curves in each section must have the same number of points.*/

```
begin section    /* Begin a new blend section.
```

This appears at the beginning of each section. */

```
begin curve      /* Begin a new curve for the section.
```

This appears at the beginning of each curve segment. */

```
1 x y z      /* The number is the point number (optional); X, Y,
```

```
2 x y z      Z are the coordinate values. */
```

```
.
```

```
.
```

```
.
```

```
# x y z
```

```
begin curve
```

```
1 x y z      /* The first point in this curve equals the last
```

```
2 x y z      point of the preceding curve. */
```

```
.
```

```
.
```

```
.
```

```
# x y z
```

```
/* end of file */
```

In this format, comments are contained between the "/* */" characters.

Notes:

- Two points in a curve define a line; more than two points define a spline.
- The endpoint of one curve and the start point of the next curve must be coincident. For closed sections, this holds true for the last point of the last curve and the first point of the first curve. There can be only one closed curve for each section and that curve must consist of at least two segments.

Sample Blend File

```

closed
arclength
begin section ! 1
  begin curve ! 1
    1      20      20      0
    2      20      30      0
    3      30      40      0

  begin curve ! 2
    1      30      40      0
    2      50      40      0
    3      60      30      0

  begin curve ! 3
    1      60      30      0
    2      60      20      0
    3      50      10      0

  begin curve ! 4
    1      50      10      0
    2      30      10      0
    3      20      20      0

begin section ! 2
  begin curve ! 1
    1      25      25      50
    2      30      30      50

  begin curve ! 2
    1      30      30      50
    2      50      25      50

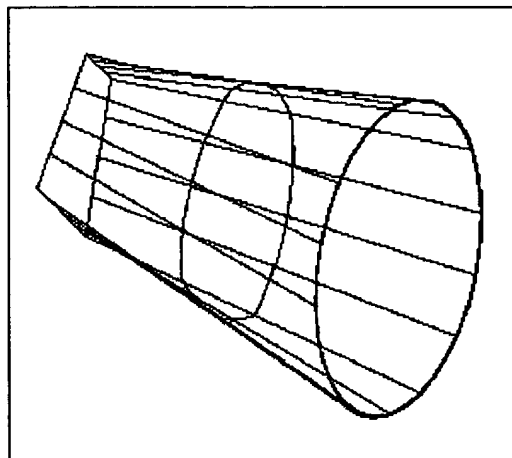
  begin curve ! 3
    1      50      25      50
    2      40      15      50

  begin curve ! 4
    1      40      15      50
    2      25      25      50

```

The following figure shows an imported blend.

Imported Blend



How to create a blend from an imported blend data points file

1. Choose **From File** from the ADV FEAT OPT menu.
2. Create or select a coordinate system that the points in the data file will reference.
3. Enter the file name. While it is not necessary to enter the extension with the filename, the file must have the extension ".ibl".
4. The system displays the first section on the model. Choose **Flip** or **Okay** to specify the direction of feature creation.
5. Pro/ENGINEER creates the blend.

Note:

When the points that are used to create a blend section from a file do not all lie on a plane, the system creates the best fit plane and projects the points down onto the plane.

How to modify a blend created from a data file

1. Choose **Modify** and pick on the blended feature.
2. Pro/ENGINEER asks if you want to edit the blend from file sections. Answer "yes".
3. Pro/ENGINEER displays a system editor window. Edit the blend file.

Note:

Modifications of a blend feature do not affect the file from which it was created. When you modify a blend, the system creates a new file, "feat_#.ibl", in your current working directory.

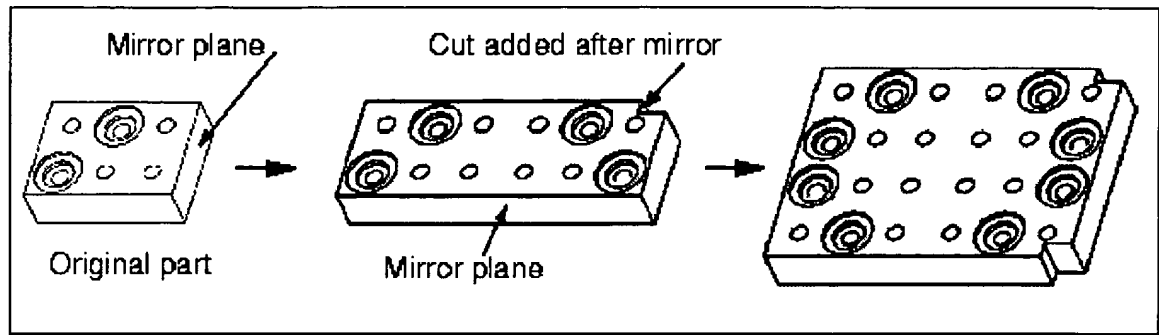
Creating a Merge Feature

All the geometry of a part can be mirrored at one time using the Mirror Geom option in the Feat menu. This creates a merge feature-the mirrored geometry merged with the original geometry. Dimensions of a merge feature cannot be shown in drawings. To show the dimensions of features that are created by mirroring and copying, create the features instead using Copy, Mirror, and All Feat (see [Copying Features by Mirror](#)).

How to mirror all geometry at one time and create a merge feature

1. Choose **Mirror Geom** from the FEAT menu.
2. Select a plane about which to mirror.

Mirroring Part Geometry



Note:

Mirroring coordinate systems always preserve the right hand rule. Pro/ENGINEER mirrors the X- and Y-axes of the coordinate system appropriately, then determines the Z-axis.

Retrieving Pro/DESIGNER Data

Use the DesignerIn option in the Feat Class menu to retrieve surface data from Pro/DESIGNER by reading in a Pro/ENGINEER surface file with the extension ".neu". Once you retrieve Pro/DESIGNER surface data, the system creates a DesignerIn feature, which becomes completely associative to other Pro/ENGINEER features-when you change the DesignerIn feature, the system updates all features that reference it.

In the Pro/ENGINEER model, you can replace the current DesignerIn feature with a modified DesignerIn feature.

Note:

Alternatively, you can transfer surface data as a DesignerIn feature directly from Pro/DESIGNER to Pro/ENGINEER, while the two applications are running on the same workstation. This method uses the Put to Pro/ENGINEER session menu option, available in Pro/DESIGNER (see the *Pro/CDRS Modeling User's Guide* for more detail).

How to create a DesignerIn feature by retrieving a surface file

1. Choose **DesignerIn** from the FEAT CLASS menu.
2. Enter the name of the surface file or type a [?] to bring up the menulist.
3. Select or create a coordinate system by using options in the GET COORDS menu. If this is the first feature in your model, the system automatically creates the default coordinate system.
4. The system reads in the file and displays the feature.

Note:

If Pro/ENGINEER displays gaps between surfaces in the DesignerIn feature, then the default Pro/DESIGNER tolerances may not be set correctly. It is recommended that you reset the Pro/DESIGNER tolerances for this model and repeat the retrieval process.

Once you create the DesignerIn feature, you can handle it as any other Pro/ENGINEER feature (for example, suppress it, put it on a layer, reorder it, and so on). To redefine the DesignerIn feature, choose Redefine and use options in the Redef Impt menu, as you do when you redefine imported surfaces (see Redefining Imported Geometry).

When you retrieve a model that has a DesignerIn feature and the system finds a more recent surface file with the same name, the system warns you that the DesignerIn feature is outdated and instructs you to regenerate the model to update that feature. If you choose Regenerate, the system ask you for a confirmation. Choose Confirm from the Confirmation menu to update the feature. Note that if you choose not update the DesignerIn feature, the system repeats the warning every time you retrieve the model.



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